



## **Quickmill Gantry Machining Centers**

***QUICKCODE*® v4.17**

**USER'S GUIDE**

**Part #09100022**



## **LICENSE**

This software is the property of QUICKMILL Inc. While QUICKMILL continues to own the software, you do have certain rights to use the software after your acceptance of this license. Your rights and obligations are as follows,

You may;

- (i) use one copy of the software on a single computer.
- (ii) copy the software onto the hard drive of your computer and store the original diskette for archival purposes.
- (iii) use the software on a network provided that you have a license for each computer that can access the software over the network.
- (iv) transfer all your rights under this license provided you retain no copies, you transfer all of the software product (diskettes and printed media) and the recipient agrees to the terms of this license.

You may not;

- (i) sub-license, rent or lease any portion of the software.
- (ii) reverse engineer, decompile, disassemble, modify, translate or create derivative products from this software.
- (iii) use a previous version of the software in addition to an upgraded version received as a replacement of the prior version.

## **WARRANTY**

QUICKMILL Inc. warrants that the media on which this software is distributed will be free from defects for a period of sixty (60) days from the date of delivery. In the event of problems with your original diskette(s) QUICKMILL Inc. will, at its option, replace any defective media returned to QUICKMILL Inc. within the warranty period. QUICKMILL Inc. makes no representations or warranties, either expressed or implied, with respect to the adequacy of this documentation or the programs it describes in regard to merchantability or fitness for any particular purpose or with respect to adequacy to produce any particular result or that the software is error free.

## **LIABILITY**

QUICKMILL Inc. does not assume any liability for any defect in the distribution media or the software, or loss of use, or for any direct, indirect, incidental, or consequential damages in connection with or arising out of the performance of the software described herein. Some states do not allow the limitation or exclusion of liability for incidental or consequential damages so the above limitation or exclusion may not apply to you.

## **COPYRIGHTS & PATENTS**

This document contains and references information and products protected by copyrights and patents and does not convey any license under the patent rights of QUICKMILL Inc., nor the rights of others.

## **REPRODUCTION**

No part of this publication may be reproduced, stored in a retrieval system, or transmitted, in any form or by any means, electronic, mechanical, photocopying, recording, or otherwise, without the prior written permission of QUICKMILL Inc. All rights reserved.

## **SERVICE POLICY**

Exceptional customer service and outstanding value are fundamental to our mission. Should you experience problems with your **QUICKMILL®** product, please contact our service department for assistance. We maintain a staff of highly trained programmers and technicians that can help you with the operation of this software.



# TABLE OF CONTENTS

|                                               |           |
|-----------------------------------------------|-----------|
| <b>CHAPTER 1</b>                              | <b>9</b>  |
| <b>OVERVIEW</b>                               | <b>9</b>  |
| PURPOSE                                       | 9         |
| FEATURES                                      | 9         |
| SYSTEM REQUIREMENTS                           | 9         |
| QUIKCODE® TERMS                               | 9         |
| COMPUTER KEYBOARD                             | 10        |
| <b>CHAPTER 2</b>                              | <b>11</b> |
| <b>INSTALLATION</b>                           | <b>11</b> |
| SINGLE USER INSTALLATION                      | 11        |
| INSTALLATION ON A QUICKMILL OR QUICKDRILL CNC | 11        |
| NETWORK INSTALLATION                          | 12        |
| <b>CHAPTER 3</b>                              | <b>13</b> |
| <b>OPERATION</b>                              | <b>13</b> |
| STARTING QUIKCODE®                            | 13        |
| PROJECT MENU                                  | 13        |
| ! Create New Project                          | 14        |
| @ Add/Edit Project                            | 14        |
| # Delete Project                              | 14        |
| % DOS Shell                                   | 14        |
| ( Select Units                                | 15        |
| ) Make Part Program                           | 15        |
| aX Quitting QUIKCODE®                         | 15        |
| PATTERN MENU                                  | 15        |
| ! From CAD File                               | 16        |
| @ Conversational                              | 16        |
| & Sort Pattern                                | 17        |
| Sorting Considerations                        | 17        |
| Sort by X                                     | 17        |
| Sort by Y                                     | 17        |
| Sorting a Pattern                             | 17        |
| * View Pattern(s)                             | 18        |
| ( Output to .DXF File                         | 19        |
| ) Make Part Program                           | 19        |

|                                               |    |
|-----------------------------------------------|----|
| v Back to Project Menu.....                   | 19 |
| k Delete Pattern.....                         | 19 |
| aX Quitting QUIKCODE® .....                   | 19 |
| CAD FILE MENU.....                            | 19 |
| ! Units .....                                 | 20 |
| @ File Type .....                             | 20 |
| # Hole Type .....                             | 20 |
| ) Back to Pattern Menu .....                  | 21 |
| k Delete File.....                            | 21 |
| aX Quit QUIKCODE® .....                       | 21 |
| CAD File Considerations.....                  | 21 |
| Hole Entity Type.....                         | 21 |
| Use of Layers or Levels .....                 | 21 |
| Location of DXF Files .....                   | 21 |
| Creating the CAD Output File .....            | 22 |
| Using the File Screen Cursor .....            | 22 |
| Deleting CAD Files .....                      | 22 |
| CONVERSATIONAL MENU .....                     | 24 |
| ! Points .....                                | 25 |
| @ Circle/Arc .....                            | 25 |
| # Line .....                                  | 26 |
| \$ Square .....                               | 27 |
| ) Back to Pattern Menu .....                  | 28 |
| TOOL MENU.....                                | 29 |
| Tool Screen.....                              | 29 |
| ! Add Tool.....                               | 29 |
| @ Edit Tool.....                              | 30 |
| k Delete Tool.....                            | 30 |
| w Back to Pattern Menu.....                   | 30 |
| x Make Part Program .....                     | 30 |
| STEP 7 - ASSEMBLING THE NC PART PROGRAM ..... | 31 |
| Make Program.....                             | 31 |
| Part Program Filename .....                   | 31 |
| ! Compustep part programs.....                | 32 |
| @ QUICKDRILL/QUICKMILL part programs .....    | 32 |
| # Fanuc 15i part programs .....               | 32 |
| \$ Fanuc 21i part programs .....              | 32 |
| Step 1: Inserting Include Files .....         | 32 |
| Step 2: Adding N Blocks .....                 | 35 |
| Step 3: Specifying Machine Data .....         | 35 |

|                                          |           |
|------------------------------------------|-----------|
| Tool Number (T code) .....               | 35        |
| Tool Length Offset Number (H code) ..... | 35        |
| Tool Description .....                   | 35        |
| Part Offset (G54 P code).....            | 36        |
| Spindle Speed (RPM) (S code).....        | 36        |
| Coolant Nozzle Type (M code) .....       | 37        |
| Feed Rate (F code).....                  | 37        |
| Canned Drill Cycle (G code) .....        | 37        |
| G89 - Auto-Circle Canned Cycle.....      | 38        |
| Layered Drill Cycle Codes .....          | 39        |
| Hole Data .....                          | 42        |
| Copy to Floppy .....                     | 42        |
| <b>CHAPTER 4</b> .....                   | <b>47</b> |
| <b>COPY FILES TO FLOPPY DISK</b> .....   | <b>47</b> |
| Copying Multiple Files .....             | 47        |
| <b>CHAPTER 5</b> .....                   | <b>49</b> |
| <b>CONFIGURATION</b> .....               | <b>49</b> |
| CONFIGURATION PARAMETERS .....           | 49        |
| 15I_DATA_DIR .....                       | 49        |
| 21I_DATA_DIR .....                       | 50        |
| ADD_HOLE_NUMS.....                       | 50        |
| B-SERIES_DATA_DIR .....                  | 51        |
| CDL_DIR.....                             | 51        |
| COOLANT_DEFAULT .....                    | 52        |
| COORDINATE_SYSTEM_ROTATION_DEFAULT ..... | 52        |
| DATA_DIR.....                            | 53        |
| DEFAULT_NEGATIVE_Z_DEPTH.....            | 53        |
| DXF_DIR.....                             | 53        |
| FANCY_SOUND .....                        | 54        |
| FLOPPY_COPY_DRIVE.....                   | 54        |
| HOLE_TYPE.....                           | 54        |
| MAX_EXTENDED_WORK_OFFSET .....           | 54        |
| MAX_RPM.....                             | 55        |
| MAX_TOOL_TABLE_NUM .....                 | 55        |
| MIN_RPM .....                            | 55        |
| OFFER_FLOPPY_COPY.....                   | 56        |
| OFFER_M41.....                           | 56        |
| OFFER_EXTENDED_WORK_OFFSETS.....         | 56        |
| PROG_DIR .....                           | 56        |

|                                   |    |
|-----------------------------------|----|
| QM_DATA_DIR .....                 | 56 |
| REALLY_SLOW_TIME .....            | 57 |
| RETURN_TO_R_IN_DRILL_CYCLES ..... | 57 |
| RIGID_TAP_SUPPORTED .....         | 57 |
| SLOW_TIME .....                   | 57 |
| TEMP_DIR .....                    | 58 |
| UNITS .....                       | 58 |
| USE_CYCLE_PRELOAD .....           | 58 |
| Undocumented Variables .....      | 59 |

---

## CHAPTER 6 61

|                                               |           |
|-----------------------------------------------|-----------|
| <b>INCLUDE FILES .....</b>                    | <b>61</b> |
| PURPOSE OF INCLUDE FILES .....                | 61        |
| CHANGING INCLUDE FILES .....                  | 61        |
| QUICKMILL/QUICKDRILL CONTROLLER .....         | 62        |
| PRE_PROG.INC .....                            | 62        |
| TC.INC .....                                  | 62        |
| END_PROG.INC .....                            | 62        |
| QUICKMILL/15I .....                           | 63        |
| 15I_PRE.INC .....                             | 63        |
| 15I_TC.INC .....                              | 63        |
| 15I_END.INC .....                             | 63        |
| INTIMIDATOR/21I .....                         | 63        |
| 21I_PRE.INC .....                             | 63        |
| 21I_TC.INC .....                              | 64        |
| 21I_END.INC .....                             | 64        |
| B-SERIES .....                                | 64        |
| B18_PRE.INC .....                             | 64        |
| B18_TC.INC .....                              | 64        |
| B18_END.INC .....                             | 65        |
| VARIABLE COORDINATES (X#.#, Y#.#, Z#.#) ..... | 65        |
| OPTIONAL LINES .....                          | 65        |

# CHAPTER 1

## OVERVIEW

---

### Purpose

**QUIKCODE®** is used to create part programs suitable for running on machining centers purchased from **QUICKMILL** Inc. Part programs can be created interactively using simple conversational menus or by converting from industry standard CAD export formats such as DXF.

**QUIKCODE®** supports part programs with drilling, tapping, boring and other "drill" type operations. The DXF file format is available on most CAD systems and provides a means of extracting hole coordinates directly from a CAD drawing. The CDL (CAD Drafting Language) file format is specific to the CADKEY program. Custom input formats can be added to **QUIKCODE®** at an additional expense.

### Features

- up to 20 patterns per project;
- up to 32767 holes per pattern;
- imports circle or point entities from a DXF or CDL file;
- supports inch or metric units;
- optional sorting of holes into rows or columns with automatic removal of duplicate holes;
- up to 12 tools per part program;
- easy to use menu-driven operation;
- fast operation;
- output to Compustep 12-bit format, **QUICKMILL/PC**, **QUICKDRILL/PC**, **QUICKMILL/15i**, **Intimidator/21i**, and Bridge Series format;
- support for all canned cycles on the intended output CNC.

### System Requirements

- IBM PC, XT, AT, or compatible computer.
- MS-DOS 3.31 or higher; **QUIKCODE®** has been tested running in a MS-DOS session on Microsoft Windows 3.1, 95, 98, NT 3.51, NT 4.0, Windows 2000, Windows XP, as well as the DOS session in OS/2 Warp 3, OS/2 Warp 4, and under the DOSEMU (DOS Emulator) in RedHat (and perhaps other) Linux distributions.
- 500K hard drive space;
- 512K RAM available for **QUIKCODE®**

### **QUIKCODE®** Terms

**part program** - a data file containing the instructions to machine a specific part.

**DXF File** - a Drawing Interchange Format file, used to exchange CAD drawings from one system to another, or extract useful information from a CAD drawing.

**CDL File** - a CAD Language file, produced by the CADKEY CAD program.

**Pattern** - a collection of holes that are produced by a common tool or set of tools.

**Project** - a collection of patterns.

**QUICKMILL/PC** - a **QUICKMILL** Gantry Machining Center with a PC-based CNC.

**QUICKDRILL/PC** - a **QUICKMILL** Drilling Machine with a PC-based CNC.

**QUICKMILL/15i** - a **QUICKMILL** Gantry Machining Center with a GE Fanuc 15i CNC.

**Intimidator/21i** - an Intimidator Machining Center with a GE Fanuc 21i CNC.

**Bridge Series** - an Eliminator or Quickdrill Bridge Series with a Fanuc 16i or 18i CNC.

## Computer Keyboard

**<enter>** - the key labeled **ENTER** key on the computer keyboard. Sometimes it is only labeled: **f**

**<esc>** - the “escape” key on the computer keyboard. This key is used throughout the program to stop operations.

**y** - the cursor up key. Moves the cursor up one line at a time.

**v** - the cursor down key. Moves the cursor down one line at a time.

**>** - the cursor right key. Moves the cursor to the right one character.

**<** - the cursor left key. Moves the cursor to the left one character.

**F1, F2, F3, F4, F5, F6, F7, F8, F9, F10** - function keys, usually located across the top of the keyboard.

**<backsp>** - the backspace key. Erases characters to the left of the cursor. Sometimes labelled Backspace.

**<del>** - the “delete” key. Erases characters to the right of the cursor. In **QUIKCODE®** it is also used to delete files, and tools.

**ALT** - the “alt” key. This key is used with other keys to issue commands to **QUIKCODE®**. For instance, the keystroke **ALT-C** means to hold down the **ALT** key and then press **C** while still holding the **ALT** key.

# CHAPTER 2

## INSTALLATION

---

**QUIKCODE**® is distributed on a single 3.5" floppy disk. **QUIKCODE**® must be installed onto the hard drive of the computer it is to be run on. It cannot be run from the distribution floppy disk.

### Single User Installation

In this installation example, **QUIKCODE**® will be installed from the A: floppy drive to the C: hard drive in the directory **C:\QC4**.

Insert the **QUIKCODE**® System Disk into the A: floppy drive.

Make the A: floppy drive the current drive. To do this, type:

A: <enter>

at the DOS prompt. The DOS prompt will change to something like:

A: \>

At the A:\> prompt, type:

INSTALL <enter>

The install routine will ask for a directory to install **QUIKCODE**®. In this example, **QUIKCODE**® will be installed in the **C:\QC4** directory, so type:

C: \QC4 <enter>

A few messages will flash by as the software is installed. Watch for any error messages. Once the installation is complete, change to the **QUIKCODE**® drive and directory by typing:

C: <enter>

CD \QC4 <enter>

and then start **QUIKCODE**® by typing:

**QUIKCODE** <enter>

The **QUIKCODE**® software will start.

### Installation on a **QUICKMILL** or **QUICKDRILL** CNC

**QUIKCODE**® can be installed on a **QUICKMILL**/PC or **QUICKDRILL**/PC since that have a PC-based CNC. This allows the CNC operator to write part programs using the features of **QUIKCODE**®

The installation routine should detect when it is installed on a **QUICKMILL**/PC or **QUICKDRILL**/PC and adjust various configuration variables accordingly. For instance, the part program path variable (**QM\_DATA\_DIR**, which tells **QUIKCODE**® where to store the resulting part programs) is adjusted to point to the **DATA** directory below the **C:\QUICKMILL** or **C:\QUICKDRILL** directory. Additionally, the **ALWAYS\_OUTPUT\_TYPE** variable will be set to **2**, which tells **QUIKCODE**® to always output part programs in **QUICKMILL**/**QUICKDRILL** G & M codes. This

saves the user from having to tell **QUIKCODE®** every time; the assumption is that if the software is installed on a **QUICKMILL/PC** or **QUICKDRILL/PC**, there is no need for outputting to the other formats **QUIKCODE®** supports. If this assumption is not correct, the **ALWAYS\_OUTPUT\_TYPE** variable can be removed from the configuration file. See **CONFIGURATION** on page 49 for details on configuration file variables.

On the **QUICKMILL/PC** or **QUICKDRILL/PC** File Menu, the **F7** key is used to load **QUIKCODE®**. If **QUIKCODE®** is not installed on the CNC, pressing **F7** will result in an error message. **QUIKCODE®** is a purchased option and is not available on all **QUICKMILL/PC** or **QUICKMILL/PC** machines. Please contact the **QUICKMILL** Inc. Sales Department for information on purchasing **QUIKCODE®**.

## Network Installation

Although **QUIKCODE®** is a single user program, it can be installed on a network file server. All users are required to have the same drive mappings to the **QUIKCODE®** directories in order to run **QUIKCODE®** from multiple workstations. Please consult **QUICKMILL** Inc. for assistance in network installations.

Note! Installing **QUIKCODE®** on a network file server does not allow for multiple users to run **QUIKCODE®**. Please contact **QUICKMILL** Inc for information on multi-user licenses.

# CHAPTER 3

## OPERATION

### Starting QUIKCODE®

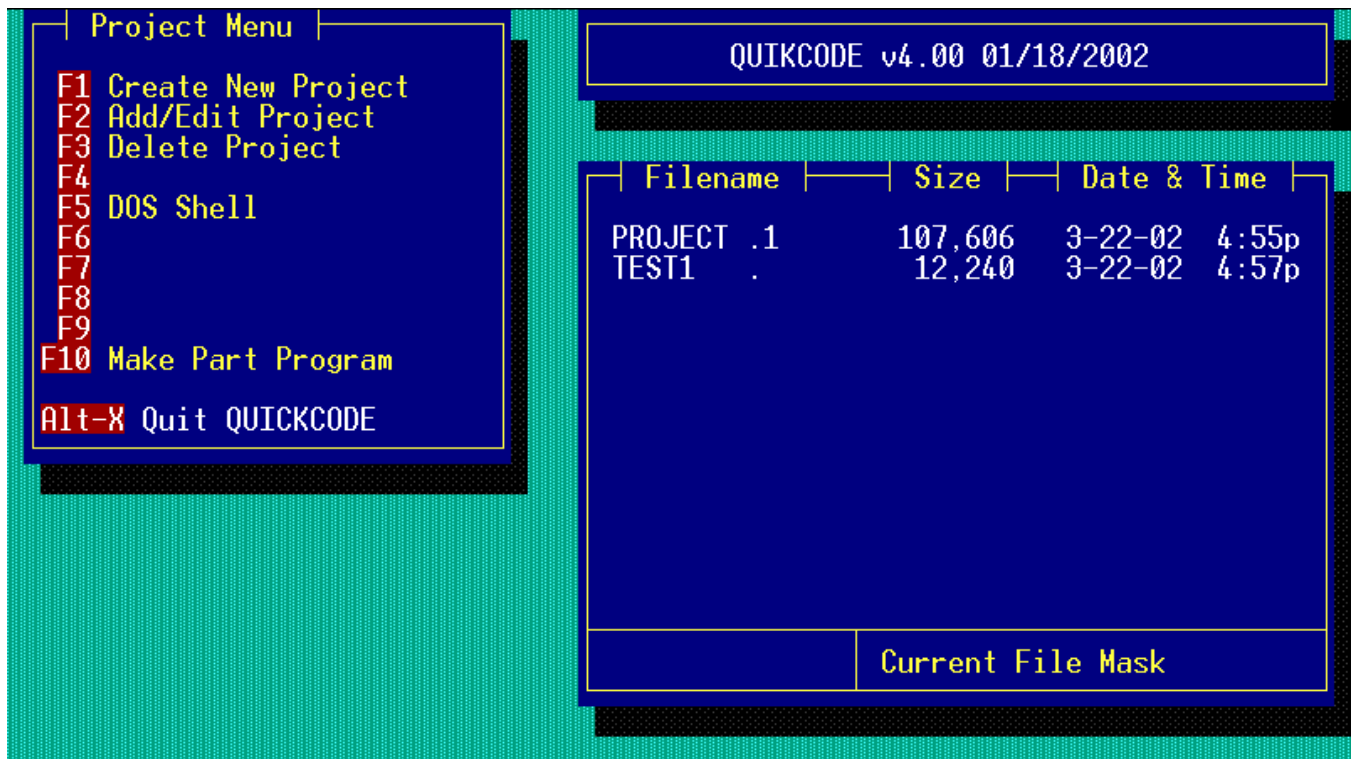
To start **QUIKCODE®**, change to the directory that contains **QUIKCODE.EXE**. At the DOS prompt, type:

**QUIKCODE <enter>**

If **QUIKCODE®** is installed on a **QUICKMILL/PC** or **QUICKDRILL/PC**, the **F7** key on the File Menu is configured to start **QUIKCODE®**

### Project Menu

The first screen displayed is the Project Menu; see *Figure #1 - Project Menu* for an example. The Project menu consists of a menu of options or commands (on the left side of the screen) and a list of project files (on the right side of the screen). Most **QUIKCODE®** screens are organized in this fashion.



**Figure #1 - Project Menu**

The files listed on the Project Menu are all the project files in the project directory, as defined in the configuration file.

The commands available on the Project Screen are:

## F1 Create New Project

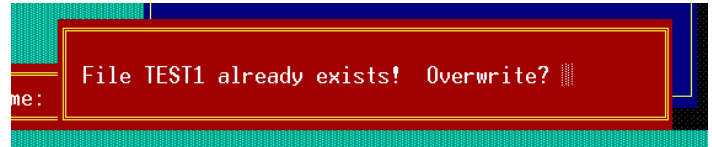
Pressing **F1** will create a new project file. **QUIKCODE®** will prompt for a filename for the new project file, as shown in *Figure #2 - Project Filename*.



**Figure #2 - Project Filename**

Enter a filename for the new project. The filename is limited to the MS-DOS 8.3 format.

If a project file with this name already exists in the project directory, **QUIKCODE®** will ask to overwrite the existing file, as shown in *Figure #3 - Overwriting an Existing Project*



**Figure #3 - Overwriting an Existing Project**

Enter **Y** to overwrite the existing project file with the same name. Enter **N** to keep the existing, in which case **QUIKCODE®** will prompt for a new filename.

WARNING! Generally, files that are overwritten files cannot be recovered. Be very careful when overwriting files.

Once a new project filename has been specified, **QUIKCODE®** changes to the Pattern Menu. See page 15 for details on the Pattern Menu.

## F2 Add/Edit Project

Pressing **F2** enables the file selection cursor. Use the **y, v, p, q, h, g** cursor control keys to highlight the project file to add to or edit. Press **<enter>** to select the highlighted file.

When **<enter>** is pressed to select a project file, **QUIKCODE®** breaks the project file into patterns and then moves to the Pattern Menu. See page 15 for details on the Pattern Menu.

## F3 Delete Project

Pressing **F3** to delete a project file. Use the **y, v, p, q, h, g** cursor control keys to highlight the project file to delete. Press **<enter>** to delete the highlighted file.

WARNING! Deleted files cannot be recovered by **QUIKCODE®**.

## F5 DOS Shell

It may be necessary to use the disk operating system of the computer while running **QUIKCODE®**. For instance, a CAD file may have to be moved or copied into the proper CAD file directory, since **QUIKCODE®** will only process files that are located in the proper directories, as defined in the configuration file.

This function is often called “shelling to DOS”. To shell to DOS from **QUIKCODE®**, press **F5**. **QUIKCODE®** will be suspended and a copy of the disk operating system will be started. The screen will look something like:

```
QUIKCODE still active. Type EXIT to return to QUIKCODE...

Microsoft(R) MS-DOS(R) Version 6.22
(C)Copyright Microsoft Corp 1981-1994.

[Windows!] C:\QC3-310>
```

**Figure #4 - The DOS Shell**

This screen will be different depending on the specific operating system on the computer. To return to **QUIKCODE®** type **EXIT** and press **<enter>**. It is very important that **QUIKCODE®** not be re-started while in the DOS shell, as this will load another copy of **QUIKCODE®**.

While in the DOS shell, the user can issue standard DOS commands to copy, move, delete files, etc. Consult your DOS manual for examples of these commands.

Note: Only persons familiar with DOS commands should use the Shell to DOS feature.

If **QUIKCODE®** is being run on a computer with a multi-tasking operating system (such as Windows or OS/2), it may be preferable to use the operating system’s file utilities (e.g. Windows File Manager or Windows Explorer) to do tasks such as copying, moving, deleting files.

## **F9Select Units**

Pressing **F9** tells **QUIKCODE®** to switch units. **QUIKCODE®** supports inch and metric modes. The current settings is listed next to the **F9** key. Pressing the **F9** key will switch to the other unit type.

On startup, **QUIKCODE®** reads the default unit type from the **UNITS** variable in the configuration file. See CONFIGURATION on page 49 for details on configuring **QUIKCODE®**.

## **F10 Make Part Program**

Pressing **F10** tells **QUIKCODE®** to make a part program from an existing project file. Use the **y**, **v**, **p**, **q**, **h**, **g** cursor control keys to highlight the project file to use. Press **<enter>** to select the highlighted file. **QUIKCODE®** will break the project file into patterns and then change to the Tool Menu. See Tool Menu on page 19.

## **aX Quitting QUIKCODE®**

To quit the **QUIKCODE®** program, press **aX**. **QUIKCODE®** will prompt the user to confirm quitting. Enter **Y** to quit, **N** to continue **QUIKCODE®**.

## **Pattern Menu**

The Pattern Menu manages the patterns in a project file. A pattern is simply a collection of **XY** coordinates that will have the same tooling operation. The Pattern Menu has functions for:

- importing patterns from CAD files such as DXF (Drafting Interchange Format) or CDL (CAD Drafting Language). Other formats may be added.
- creating patterns interactively or conversationally by answering questions.
- sort patterns for more efficient processing.
- delete patterns.

See *Figure #5 - Pattern Menu* for an example of the Pattern Menu. The Pattern Menu consists of a menu of options or commands (on the left side of the screen) and a list of the patterns in the current project file (on the right side of the screen). If this is new project file (created with the **F1** key on the Project Menu), the list of patterns will be empty.

| Pattern Menu |                        | Pattern Name | Sort | Holes |
|--------------|------------------------|--------------|------|-------|
| F1           | From CAD File          | CIRCLE1      |      | 24    |
| F2           | Conversational         | CIRCLE2      |      | 24    |
| F3           |                        | CIRCLE3      |      | 24    |
| F4           | Sort Pattern           | CIRCLE4      |      | 25    |
| F5           | Delete Pattern         | 2-THOLE      |      | 19    |
| F6           |                        |              |      |       |
| F7           |                        |              |      |       |
| F8           |                        |              |      |       |
| F9           | Output to .DXF File    |              |      |       |
| F10          | Return to Project Menu |              |      |       |

**Figure #5 - Pattern Menu**

The list of patterns shows the name, sort order, and number of holes in the pattern. If a pattern has not been sorted, the **Sort** column will be blank.

The commands available on the Pattern Menu are:

### **F1 From CAD File**

Pressing **F1** will switch to the CAD File Menu. See *CAD File Menu* on page 19 for details on the CAD File Menu.

### **F2 Conversational**

Pressing **F2** will cause **QUIKCODE®** to switch to the Conversational Menu. The Conversational Menu allows for adding patterns by answering questions and providing values for points, circles, arcs, lines of holes, etc. See *Conversational Menu* on page 24 for details of the Conversational Menu.

## F7 Sort Pattern

Each pattern can be sorted into alternating rows or columns. Pressing **F7** will allow selecting of a pattern for sorting.

### Sorting Considerations

Each pattern can be sorted into alternating rows in the **X** or **Y** axis to minimize the motion of the tool. Patterns that are constructed in the CAD software using **ARRAY**, **MIRROR**, or **COPY** commands usually need sorting. Most CAD software does not order entities in an efficient, logical manner, at least as far as tool paths are concerned.

### Sort by X

Sorting by the **X** axis will produce a part program with the holes organized into rows in the **X** axis. The **Y** axis will complete a row before the **X** axis is moved to the next row. The direction of the **Y** axis will be reversed for each row, resulting in the quickest tool path. This is the preferred sorting method for machines that position in the **Y** axis faster than the **X** axis.

### Sort by Y

Sorting by the **Y** axis will produce a part program with the holes organized into rows in the **Y** axis. The **X** axis will complete a row before the **Y** axis is moved to the next row. The direction of the **X** axis will be reversed for each row, resulting in the quickest tool path. This is the preferred sorting method for machines that position in the **X** axis faster than the **Y** axis.

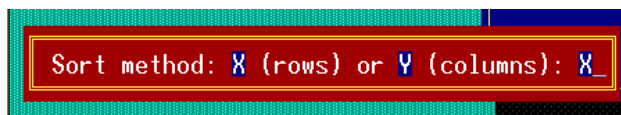
Certain patterns should not be sorted. Circular patterns of holes are best left unsorted, since sorting them into rows will result in needlessly long tool travel.

Patterns are sorted on an individual basis. For example, if there are two patterns in the layout, one, both, or none of the patterns can be sorted.

### Sorting a Pattern

To sort a pattern, press **F7** then use the **^** or **v** cursor keys to highlight the pattern to be sorted, then press **<enter>** to select it. If only one pattern exists, **QUIKCODE®** will automatically select it. **QUIKCODE®** will then prompt for the sort type:

Enter **X** or **Y** as required. Sorting is a four phase process. First, the pattern is read into the computer's memory from disk. Then there are two sorting phases. Finally the pattern is written back to disk. While the pattern is written back to disk, any duplicated holes will be removed. The sorting process will take a very short time on modern Pentium-class computers, even for very large patterns. For example, a test of a 20,000 hole circular pattern can be sorted in under 5 seconds on a Pentium III/833 Mhz computer.



**Figure #6 - Choosing a Sort Method**

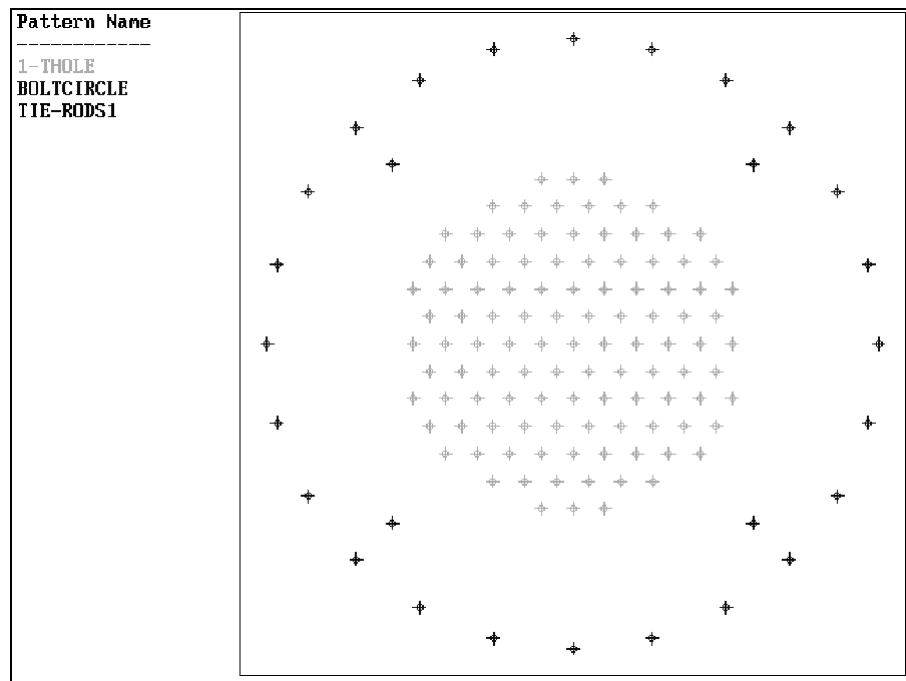
When sorting is completed, the number of holes in the pattern is updated in the table to indicate the total number of holes in the pattern. Duplicate holes, where more than one CAD entity is located at the same **XY** coordinates, will have been removed. If there is still a discrepancy between the holes per pattern shown and the actual holes to drill, then there must have been some unwanted hole type

entities (**CIRCLE** or **POINT**) in the CAD file. Check the original drawing using the CAD program to verify the number of hole entities present and remove the extra hole entities.

## F8 View Pattern(s)

For testing purposes, it may be convenient to view the patterns in a graphical manner. If there is only one pattern defined, **QUIKCODE**® will plot that pattern when the **F8** key is pressed. If more than one pattern exists, **QUIKCODE**® will prompt **Plot all patterns?**, with **Y** (for yes) as the default. If **N** (for no) is entered, **QUIKCODE**® will ask the operator to use the **^ v** cursor keys to select the pattern to view. Press **<enter>** to plot the selected pattern.

The patterns will be plotted at a scale such that all holes will fit comfortably on the screen. Since **QUIKCODE**® does not know the size of the hole (at the pattern level, all **QUIKCODE**® knows about each hole is its **XY** coordinates), a suitable size is chosen relative to the overall scale of the pattern(s). See *Figure #7 - Pattern Plot Example* for an example plot.



**Figure #7 - Pattern Plot Example**

**QUIKCODE**® will assign a different colour to each pattern. The names of the patterns plotted and their assigned colours is listed on the left side of the screen.

While the patterns are being plotted, the following keys are active:

- holding the Left **<SHIFT>** key will slow the plotting by 0.1 seconds per hole.
- holding both the Left & Right **<SHIFT>** keys will slow the plotting by 0.5 seconds per hole.

The time delays during plotting are set in the **QUIKCODE**® configuration file via the **SLOW\_TIME** and **REALLY\_SLOW\_TIME** variables. See *CONFIGURATION* on page 49 for details on setting configuration file variables.

Once the pattern(s) have been plotted, **QUIKCODE**® will accept the following keys:

- press **<ESC>** to leave the plot mode.
- press the **+** key to zoom in 10%.
- press the **-** key to zoom out 10%.
- press the **A** key to automatically scale the plot to full screen.

## F9 Output to .DXF File

For testing purposes, it may be convenient to output the patterns to a DXF file, suitable for importing into CAD software or other software capable of viewing DXF files. Pressing **F9** will bring up a simple dialog box asking for the DXF filename. Enter a filename in MS-DOS 8.3 format. If the **.DXF** extension is not specified, **QUIKCODE®** will automatically add the **.DXF** extension to the filename.

The patterns will be written to the specified filename in the directory setup by the **DXF\_OUT\_DIR** setting in the configuration file. This is typically the directory **DXF\_OUT** below the **QUIKCODE®** install directory.

The DXF file output with this function is a very simple file. It has been tested with AutoCAD r14, AutoCAD 2002, and CADKEY 98. Due to the ever-changing nature of the DXF format, **QUICKMILL** cannot guarantee that the DXF file output from **QUIKCODE®** can be imported into every possible DXF-capable application.

## F10 Make Part Program

Pressing **F10** tells **QUIKCODE®** to make a part program from an existing patterns. **QUIKCODE®** will change to the Tool Menu. See Tool Menu on page 19.

## V Back to Project Menu

Pressing **F10** will close the Pattern Menu and return to the Project Menu. All changes are automatically saved to the current project file.

## <del> Delete Pattern

The **<del>** Delete Pattern function allows pattern(s) to be removed from the project. To delete a pattern, press **<del>** then use the **y** or **v** cursor keys to highlight the pattern to be deleted, then press **<enter>** to delete it. **QUIKCODE®** will prompt the operator to confirm the deletion.

## aX Quitting QUIKCODE®

To quit the **QUIKCODE®** program, press **aX**. **QUIKCODE®** will prompt the user to confirm quitting. Enter **Y** to quit, **N** to continue using **QUIKCODE®**.

## CAD File Menu

The CAD File Menu is used to import patterns from files made by CAD programs. **QUIKCODE®** supports the DXF (Drawing Interchange Format), CDL (CAD Drafting Language). Part programs in the Compustep 12-bit format can also be imported. Other formats may be added on a customer-request basis.

The CAD File Menu consists of a list of functions on the left side of the screen and a list of files on the right. See Figure #8 - CAD File Menu for an example of the CAD File Menu. The following

| CAD File Menu |                      |         | Filename     | Size   | Date & Time |       |
|---------------|----------------------|---------|--------------|--------|-------------|-------|
| F1            | Units                | ENGLISH | SAMPLE .DXF  | 69,544 | 5-31-96     | 3:10a |
| F2            | File Type            | *.DXF   | TUBESHT1.DXF | 28,810 | 5-31-96     | 3:10a |
| F3            | Hole Type            | CIRCLE  |              |        |             |       |
| F4            |                      |         |              |        |             |       |
| F5            |                      |         |              |        |             |       |
| F6            |                      |         |              |        |             |       |
| F7            |                      |         |              |        |             |       |
| F8            |                      |         |              |        |             |       |
| F9            |                      |         |              |        |             |       |
| F10           | Back to Pattern Menu |         |              |        |             |       |
| DEL           | Delete File          |         |              |        |             |       |

**Figure #8 - CAD File Menu**

functions are available on the CAD File Menu:

### **F1 Units**

Pressing **F1** will switch the units *QUIKCODE*<sup>®</sup> uses, either **INCH** or **METRIC**. This is required since some CAD file formats do not explicitly define the units.

The default type of units is set by the **UNITS** variable in the *QUIKCODE*<sup>®</sup> configuration file. See page 49 for details on configuring *QUIKCODE*<sup>®</sup>.

### **F2 File Type**

Pressing **F2** selects what type of CAD file to work on. Pressing **F2** switches between **DXF** and **CDL**; if additional CAD file formats have been created, they will also be listed. Each time the **File Type** is changed, the list of files is updated to show just the type of file specified.

*QUIKCODE*<sup>®</sup> looks for CAD files in directories set in the *QUIKCODE*<sup>®</sup> configuration file. CAD files must be copied to these directories before they are visible to *QUIKCODE*<sup>®</sup>. See page 49 for further details on setting the CAD file directories.

The default CAD file type is set by the **CAD\_FILE\_TYPE** variable in the *QUIKCODE*<sup>®</sup> configuration file. See page 49 for details on configuring *QUIKCODE*<sup>®</sup>.

### **F3 Hole Type**

Pressing **F3** key selects the type of CAD entity *QUIKCODE*<sup>®</sup> will look for in the CAD file. Pressing **F3** switches between **CIRCLE** and **POINT**. When processing the CAD file, every time an entity of this

type is encountered, the X & Y coordinates will be extracted and **QUIKCODE**® will consider this as a “hole” location.

The default type of entity to look for is set by the **HOLE\_TYPE** variable in the **QUIKCODE**® configuration file. See page 49 for more information on setting options in the configuration file.

## **F10 Back to Pattern Menu**

Pressing **F10** closes the CAD File Menu and returns to the Pattern Menu.

## **<del> Delete File**

CAD files can be deleted by pressing the **<del>** key.

## **aX Quit QUIKCODE®**

To quit the **QUIKCODE**® program, press **aX**. **QUIKCODE**® will prompt the user to confirm quitting. Enter **Y** to quit, **N** to continue **QUIKCODE**®

## **CAD File Considerations**

When creating a CAD drawing, the following points will make translating the CAD file easier.

### **Hole Entity Type**

**QUIKCODE**® can look for either **CIRCLE** or **POINT** entities for the location of each hole, but not both in the same CAD file.

### **Use of Layers or Levels**

By using the layer or level capability of your CAD system, a separate drawing for each set of hole patterns is not required. When **QUIKCODE**® reads in a CAD file it will separate the holes into patterns based on the layers or levels defined in the CAD system. For this reason, the CAD drawing should have patterns of holes separated into layers. All the holes that will have a common drilling operation should be on the same layer. Name the layers accordingly, if your CAD system allows named layers.

For instance, a typical tube sheet may have holes for the tubes and holes for a bolt circle. This sort of drawing is best laid out with all the tube holes on one layer and all the bolt holes on a different layer. If your CAD system allows for named layers, the layer with the tube holes could be named **TUBE\_HOLES** and the layer with the bolt holes could be named **BOLT\_CIRCLE**.

Before using the output function of your CAD system, all layers or levels that do not contain holes should be turned off. This makes the CAD output file smaller, reducing processing time. It also ensures that entities that are not part of the hole patterns are not included in the CAD output file.

### **Location of DXF Files**

**QUIKCODE**® will only process CAD files from a specific directory on the hard drive, set in the configuration file. If your CAD system allows full drive and path names, this will be no problem. However, on CAD systems that write output files to a fixed directory, the DOS **COPY** command can be used to copy the CAD output files to the **QUIKCODE**® directory. Alternatively, **QUIKCODE**® can be configured to look for CAD files in the CAD program's directory.

See *CHAPTER 5 - QUIKCODE® Configuration* on page 49 for more information on setting the directories for the CAD files.

### Creating the CAD Output File

Use the output function of your CAD program to write a DXF or CDL file. The DXF output format is available from most CAD programs. CDL output is typically only available from CADKEY. Consult your CAD system's manual for details on writing DXF or CDL files.

If your CAD system has a “binary DXF” output available, **DO NOT** use it. **QUIKCODE®** does not support binary DXF files.

If possible, turn off all layers or levels that do not contain holes before writing the output file.

When using the output function of your CAD program, it is a good idea to select only the entities that correspond to holes. This greatly reduces the size and complexity of the CAD file.

### Using the File Screen Cursor

The cursor on the File List is controlled with the following keys:

- y** moves the list of files up one line.
- v** moves the list of files down one line.
- p** moves the list of files up one page (13 lines)
- q** moves the list of files down one page (13 lines)
- h** moves to the top of the list of files
- g** moves to the bottom of the list of files

In order to perform an action on a CAD file, it must first be highlighted with the cursor.

### Deleting CAD Files

The CAD file highlighted by the cursor can be deleted from the File List. This is useful when a CAD file is no longer required.

To delete a CAD file, put the cursor on the CAD file and press the **<del>** key. **QUIKCODE®** will prompt the user to confirm that the indicated CAD file should be deleted. Enter **Y** to tell **QUIKCODE®** to delete file, otherwise, enter **N** to keep the file.

NOTE: Once a file is deleted, it is very difficult, if not impossible, to restore the file once it has been deleted. Users should be very careful when using the **<del>** key!

If **QUIKCODE®** cannot find any entities of the current type, as set by the **Hole Type** setting on the File Screen, **QUIKCODE®** will alert the user and ask if it should re-process the CAD file using the other type of entity.

The Sort Screen lists the current **Units** and **Hole Type** for informational purposes only. These settings cannot be changed on the Sort Screen -- they are set in the File Screen. The **Input File** field will now show the complete filename of the CAD file that was processed.

The right side of the Sort Screen contains a list of the patterns that **QUIKCODE®** found in the CAD file. The patterns will be listed in the order they are found in the CAD file. See Figure #9 for an example.

| CAD File Menu |                      |         | Filename     | Size   | Date & Time |       |
|---------------|----------------------|---------|--------------|--------|-------------|-------|
| F1            | Units                | ENGLISH | SAMPLE .DXF  | 69,544 | 5-31-96     | 3:10a |
| F2            | File Type            | *.DXF   | TUBESHT1.DXF | 28,810 | 5-31-96     | 3:10a |
| F3            | Hole Type            | CIRCLE  |              |        |             |       |
| F4            |                      |         |              |        |             |       |
| F5            |                      |         |              |        |             |       |
| F6            |                      |         |              |        |             |       |
| F7            |                      |         |              |        |             |       |
| F8            |                      |         |              |        |             |       |
| F9            |                      |         |              |        |             |       |
| F10           | Back to Pattern Menu |         |              |        |             |       |
| DEL           | Delete File          |         |              |        |             |       |

**Figure #9 - CAD File Menu**

## Patterns Name

The **Patten Name** column lists the names of the layers or levels, as defined in the CAD file, that **QUIKCODE®** found in the CAD file. Only layers that contain entities of the type selected by the **Hole Type** setting are included. In this example, 4 layers were found in this CAD file. The layers are **TIE-RODS**, **TUBES**, **BOLT\_CIRCLE**, and **SHEET**.

Note that some CAD programs are not capable of descriptive layer names as in this example. In those cases, the layers may be simply numbers. If your CAD program is able to use descriptive names for layers, it is a good idea to give layers a meaningful name. This reduces confusion when processing the CAD file into a part program.

## Sort

If a pattern has been sorted, the **Sort** column will contain either an **X** or a **Y** to indicate which axis was used for the sort. If the pattern has not been sorted, this column will be blank. Sorting is described later in this manual.

## Holes

For each layer, **QUIKCODE®** lists the total number of “holes” found in the CAD file. **QUIKCODE®** keeps a running count of the total number of holes it has found in the CAD output file in the **Total** field. As **QUIKCODE®** discovers each layer or level in the CAD file it will list them in the **Pattern Name** column. Any layer or level that contains an entity of the current **Hole Type** (**POINT** or

**CIRCLE**) will be displayed in this table. The activity indicator will continue to spin as the CAD output file is processed to alert the operator that **QUIKCODE®** is working on the file. On very large CAD files, or on slower computers, there may be extended periods of processing where no patterns are found. As long as the activity indicator is still spinning, the CAD file is being processed.

NOTE: Depending on how the entities were drawn in the CAD software, **QUIKCODE®** may find more holes than were intended for each pattern. This is usually caused by using **ARRAY** and/or **MIRROR** commands (or equivalent commands) when drawing the holes in the CAD software. Depending on how these commands are used, there may be holes that have more than one **POINT** or **CIRCLE** in the same **XY** location. **QUIKCODE®** will automatically remove any duplicate holes found in the sorting phase.

## Conversational Menu

The Conversational Menu is used to create patterns by answering questions to describe the patterns in terms of simple shapes (single points, circles, arcs, lines and squares). The allows for the creation of part programs directly from a print when CAD software is not available.

Before switching to the Conversational Menu, **QUIKCODE®** needs to know if the pattern is a new pattern or if an existing pattern is to be used. Before the Conversational Menu is shown, **QUIKCODE®** will prompt:

If an existing pattern is to be used, the **y** or **v** cursor keys can be used to select the existing pattern.

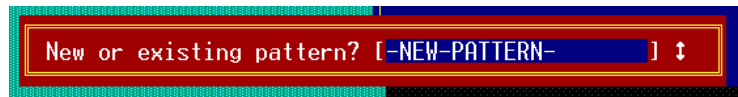


Figure #10 - CAD File Menu

If this is a new pattern, press **<enter>** on the **-NEW PATTERN-** entry; **QUIKCODE®** will then prompt for a name to give the new pattern:

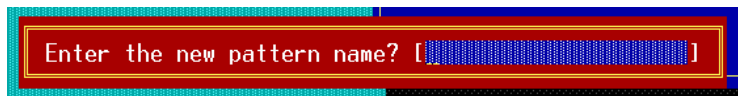
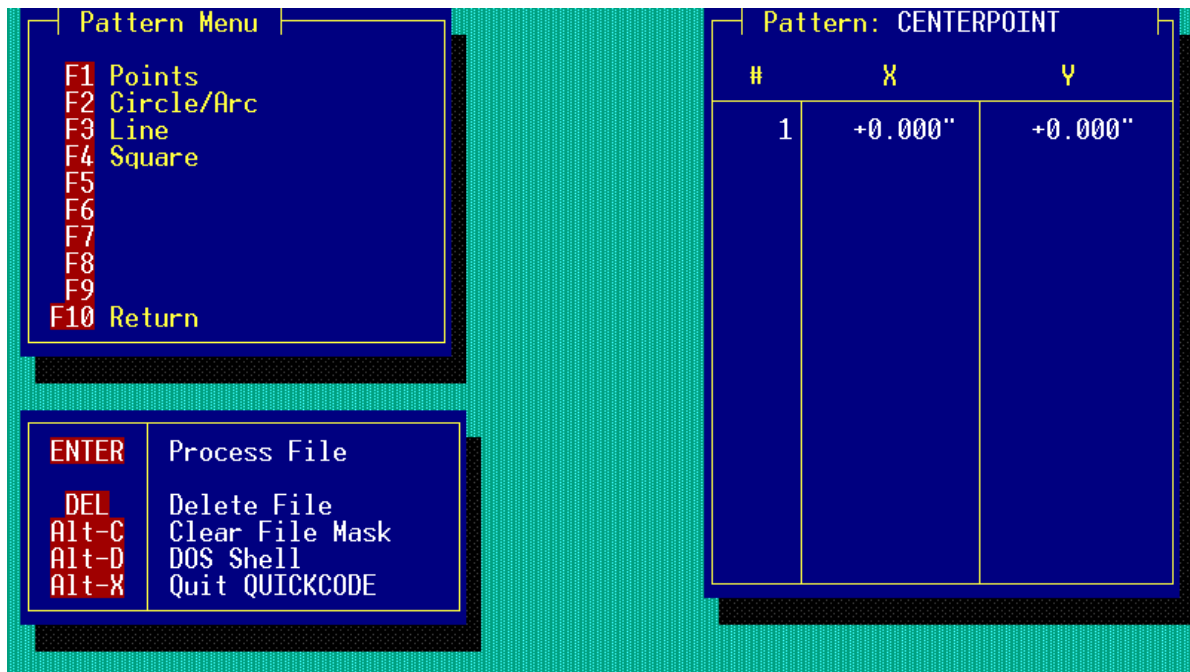


Figure #11 - CAD File Menu

Enter a name for the new pattern. If the name of the pattern already exists, **QUIKCODE®** will not accept the name. If no pattern name is entered or if **v** is pressed, **QUIKCODE®** will return to the Pattern Menu.



**Figure #12 - Conversational Menu**

The Conversational Menu consists of a list of functions on the left side of the screen and a list of the coordinates in the current pattern on the right. See *Figure #12 - Conversational Menu* for an example.

The following functions are available on the Conversational Menu:

### **F1 Points**

Pressing the **F1** key will prompt for the **X** and **Y** coordinates for single point(s), as shown in *Figure #13 - Entering a Point*



**Figure #13 - Entering a Point**

Enter the **X** coordinate of the point then press **<enter>**. **QUICKCODE®** will then prompt for the **Y** coordinate of the point:

Enter the **Y** coordinate of the point then press **<enter>**. **QUICKCODE®** will add the the point to the list and will prompt for the next **X** and **Y** coordinates. **QUICKCODE®** will continue prompting for **X** and **Y** coordinates until a blank entry is input or the **v** key is pressed.

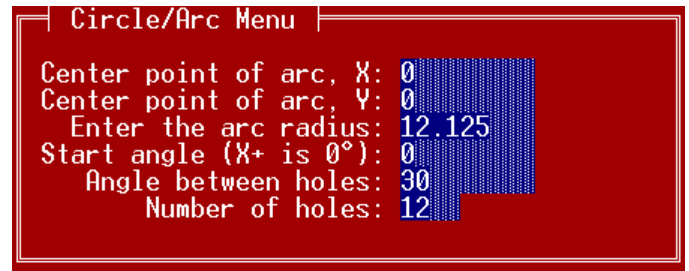
### **F2 Circle/Arc**

The **F2 Circle/Arc** function allows for the creation of circular patterns of holes. **QUICKCODE®** will prompt for the information that describe the circular pattern, in the following order:

1. Center point of arc, **X**:
2. Center point of arc, **Y**:
3. **Radius** or **Diameter**:

The next prompt is to determine how the diameter of the arc or circle will be specified. **QUIKCODE®** accepts either radius or diameter; enter **R** or **D** to select the type.

The next prompt will either ask for the arc radius or diameter. In *Figure #14 - Arc/Circle Entry* the radius is being prompted for.



**Figure #14 - Arc/Circle Entry**

#### 4. Start Angle

The next prompt is for the start angle of the first hole in the arc. This origin of this angle (the zero degree angle) is a line from the center of the arc pointing in the **X+** direction. Enter an angle from 0 to 360 degrees.

#### 5. Angle Between Holes

The next prompt is the angle between the holes on the arc. Positive angles are counter-clockwise, negative angles are clockwise.

#### 6. Number of Holes

The final prompt is for the number of holes on the arc. For convenience, if the angle between holes is evenly divisible into a full circle, **QUIKCODE®** calculates the total number of holes and offers this as a default. In *Figure #14*, **QUIKCODE®** is offering **12** for the total number of holes, since  $360 / 30 = 12$ .

If this default is not correct, simply press the **<backsp>** key to erase the default and enter the correct number.

**QUIKCODE®** will then calculate all the holes in the arc or circle. The coordinates of each hole will be listed; if there are more holes than can be listed on the screen, **QUIKCODE®** will pause and prompt the operator to press **<enter>** to list the coordinates one screen at a time, or press **A** to have **QUIKCODE®** list all the coordinates at once.

Once all the coordinates are listed, the Conversational Menu is displayed again. Further holes can be added or the pattern can be closed and returned to the Pattern Menu.

## F3 Line

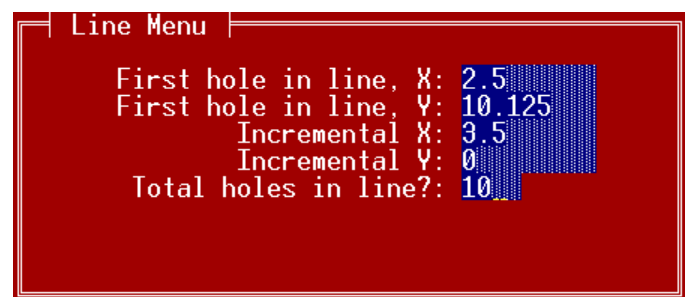
The **Line Menu** is accessed by pressing the **F3** key. Lines are described by specifying the first hole in the line and either the incremental distance from the first hole to the second or the angle and distance to the second hole. The number of similarly-spaced holes is then specified. See for an example.

#### 1. First Hole in Line, X and Y

**QUIKCODE®** prompt for the X and Y coordinates as in *Figure #15 - Line with Incremental XY*:

#### 2. Incremental XY or Vector

**QUIKCODE®** allows for specifying the distance between holes in a line by one of two methods:



**Figure #15 - Line with Incremental XY**

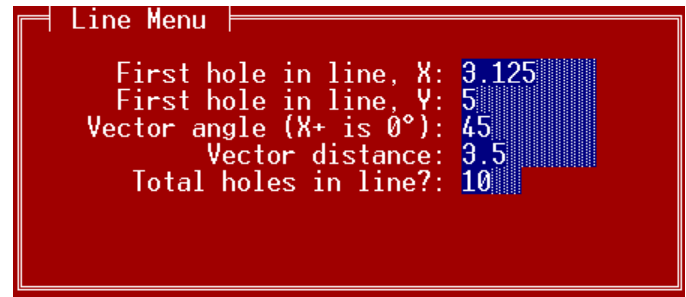
- a) **Incremental XY**: the distance in X and Y from the first hole to the second hole in the line. See *Figure #15* for an example of the incremental XY method.
- b) **Vector**: the angle and distance from the first hole to the second hole. See *Figure #16* for an example of the vector angle method.

### 3. Number of Holes

The total number of holes, including the first, is now entered.

Once the required information is entered **QUIKCODE®** will calculate all the holes in the line. The coordinates of each hole will be listed; if there are more holes than can be listed on the screen, **QUIKCODE®** will prompt the operator to press **<enter>** to list the coordinates one screen at a time, or press **A** to have **QUIKCODE®** list all the coordinates at once.

Once all the coordinates are listed, the Conversational Menu is displayed again. Further holes can be added or the pattern can be closed and returned to the Pattern Menu.



**Figure #16 - Line with Vector Angle**

## F4 Square

Square patterns of holes can be programmed with the **F4 Square** menu option. A square pattern is defined much like a line; a baseline is created just like a single line, and then this line is repeated at a specified offset. The interior of the resulting square can be filled with holes or left empty.

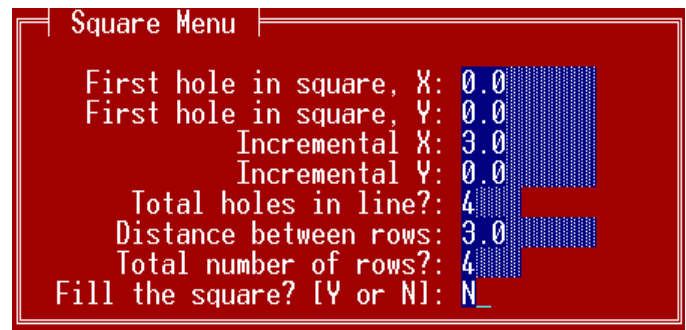
### 1. First Hole in Square, X and Y

**QUIKCODE®** will prompt for the X & Y coordinates of the first hole in the square pattern. The first hole is typically the hole in the lower left corner of the square pattern, although it is possible to use any corner hole in the square pattern as the “first hole”.

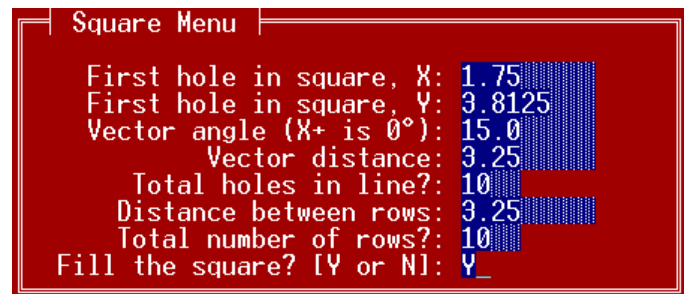
### 2. Incremental or Vector

The location of the second hole in the baseline can be determined by an incremental X and Y distance or by a vector describing the angle and distance from the first hole to the second. **QUIKCODE®** prompts the operator which type will be used; enter **I** for incremental X & Y or **V** for vector.

- a) If Incremental X & Y is used,



**Figure #17 - Square Pattern, Incremental XY**



1.

### 2. Figure #18 -

**QUIKCODE®** prompts for the distances in X and Y.

- b) If Vector is chosen, **QUIKCODE®** prompts for the angle of the vector (the angle from the first hole to the second). The origin of this angle (the zero degree angle) is a line from the center of the first hole pointing in the X+ direction. Enter an angle from 0 to 360 degrees. **QUIKCODE®** will then prompt for the vector distance from the center of the first hole to the center of the second hole.

3. Total Holes in Line

To complete the baseline of the square, **QUIKCODE®** needs to know the total number of holes in the line. Enter the total number of holes in the line, **including the first hole**. The minimum number of holes is 2, since a line of 1 hole is better known as a point, and a line with 0 or less holes is just plain wrong.

4. Distance Between Rows

Once a baseline has been established for the square, **QUIKCODE®** needs to know the distance between the rows. This is a single dimension (not an X & Y pair), since the angle of the baseline row has already been defined.

5. Total Number of Rows

**QUIKCODE®** now needs to know the total number of rows in the square pattern. The baseline row will be repeated this number of times at the offset distance between each row. Enter the total number of rows **including the baseline**. The minimum number of rows is 2, since 1 row does not make a square, and 0 or less rows is just plain wrong.

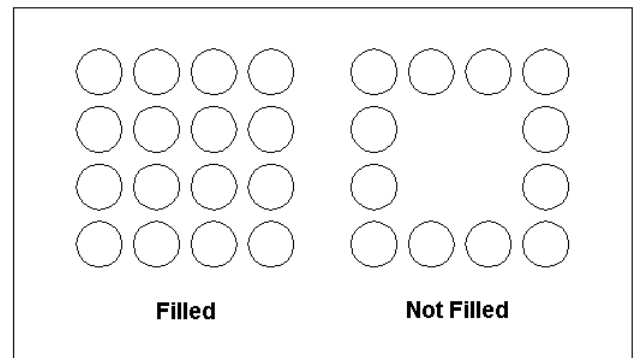
6. Fill the Square?

Finally, **QUIKCODE®** needs to know if the interior of the square should be filled or not. If the square is not filled, only the perimeter of the square will have holes. See Figure #19 for an example of filled vs. non-filled square patterns.

Once all the information has been entered **QUIKCODE®** will calculate all the holes in the square pattern. The coordinates of each hole will be listed; if there are more holes

than can be listed on the screen, **QUIKCODE®** will prompt the operator to press **<enter>** to list the coordinates one screen at a time or press **A** to have **QUIKCODE®** list all the coordinates at once.

Once all the coordinates are listed, the Conversational Menu is displayed again. Further holes can be added or the pattern can be closed and returned to the Pattern Menu.



**Figure #19 - Filled and Not Filled Squares**

Once all the coordinates are listed, the Conversational Menu is displayed again. Further holes can be added or the pattern can be closed and returned to the Pattern Menu.

## **F10 Back to Pattern Menu**

Pressing **F10** will close the current pattern and return to the Pattern Menu. The newly added pattern will be at the bottom of the list of patterns.

## Tool Menu

### Tool Screen

The Tool Screen is used to define the tools to be used in the part program, and the patterns of holes to use each tool on.. See Figure #20 for an example Tool Screen. The Tool Screen commands are listed in the table in the lower left of the screen.

| Tool   | Size   | Depth   | Rapid Z | Pattern     |
|--------|--------|---------|---------|-------------|
| DRILL  | 1.125" | -2.500" | 0.500"  | TUBES       |
| REAMER | 1.135" | -2.500" | 0.500"  | TUBES       |
| DRILL  | 2.250" | -2.750" | 0.500"  | BOLT_CIRCLE |

|     |                      |
|-----|----------------------|
| F1  | Add Tool             |
| F2  | Edit Tool            |
| DEL | Delete Tool          |
| →   | Make Program         |
| ←   | Back to Pattern Menu |

**Figure #20 - Tool Screen**

### F1 Add Tool

To add a tool to the part program, press **F1**. The word **DRILL** will appear in the table for the next tool. Press the **y** or **v** to cycle to the next tool. When the correct tool is displayed, press the **<enter>** key.

There are six tools defined in **QUIKCODE®**: **DRILL**, **REAMER**, **TAP**, **COUNTERSINK**, **HOLEGROOVER**, and **OTHER**. These tools only have meaning when the part program is to be written in Compustep 12-bit format. When the part program is written in **QUICKDRILL/PC**, **QUICKMILL/PC**, or **QUICKMILL/Fanuc 15i** format, this tool type information is replaced by tool number (T word), tool length offset (H word) and canned drill cycle information.

**QUIKCODE®** now requests the tool size. Enter the size, in the current units. Press the **<enter>** key when done.

**QUIKCODE®** next requests the tool penetration depth. Enter the depth, in current units. Press the **<enter>** key when done. This value will become the **Z** word in **QUICKMILL/PC**, **QUICKDRILL/PC**, and **QUICKMILL/Fanuc 15i** format part programs.

Note: For Compustep 12-bit format part programs, the tool penetration depth must be a positive number. For **QUICKMILL/PC**, **QUICKDRILL/PC**, or Fanuc CNCs, the tool penetration depth is usually a negative number, although it may be positive, depending on the part & tool length offsets used in the part program.

**QUIKCODE®** next prompts for the “Rapid Z” value for this tool. The Rapid Z value is the distance to raise the tool from the work piece when traversing from hole to hole. Enter the Rapid Z value, in

current units. Press the **<enter>** key when done. This value will become the R word in *QUICKMILL/PC*, *QUICKDRILL/PC*, and *QUICKMILL/Fanuc 15i* format part programs.

Note: The Rapid Z value for both Compustep 12-bit format part programs must be positive. For *QUICKMILL/PC*, *QUICKMILL/PC*, or Fanuc CNCs, the Rapid Z value (which will appear in the part program as the R word on the drill cycle line) is usually a positive number, although it depends on the part & length offsets in the part program.

**QUICKCODE**® now asks for the pattern of holes that this tool should operate on. The first pattern will be displayed. Press the **y** or **v** to cycle through the available patterns. Press the **<enter>** key when the appropriate pattern is displayed.

Note: If there is only one pattern available, it will automatically be entered into the Pattern column in the table.

While entering any value for a tool, pressing the **v** key will abort the tool entry process and remove the tool from the list.

A total of twelve tools may entered in the Tool Screen.

## **F2 Edit Tool**

Edits an existing tool specification. When the **F2** key is pressed, a cursor will appear in the **Tool** column. Use the **y** and **v** keys to move to the tool to be edited, then press **<enter>** to select it for editing. **QUICKCODE**® will then prompt for the tool data as if a new tool is being entered, but each prompt will use the current value for the default. If a column does not have to be changed, simply press **<enter>** to use the default.

### **<del> Delete Tool**

To remove a tool from the Tool Screen, press the **<del>** key. Use the **y** and **v** keys to move to the tool to be deleted, then press **<enter>** to delete the tool.

### **< Back to Pattern Menu**

To move back to the Pattern Menu, press the **<** key. This will allow the user to sort a pattern that may have been left unsorted. Tool information already entered will be preserved.

### **> Make Part Program**

When all the tool information has been entered for each pattern, the part program can be made. Press the **>** key to start the steps required to make the part program.

## Step 7 - Assembling The NC part program

### Make Program

Once all the tools have been specified and assigned a pattern, the part program can be made. **QUICKCODE®** will now request what format to use for the output part program. There are three different part program formats::

**Compustep**.....the original moving table 60” x 60” Compustep Drilling Machines.

**QUICKMILL/QUICKDRILL**.....machines with a PC-based **QUICKMILL** control.

**Fanuc 15i**.....machines with a Fanuc 15i CNC.

**Fanuc 21i**.....machines with a Fanuc 21i or 0i CNC.

**B-Series**.....bridge-style Eliminator or Quickdrill with a Fanuc 16i or 18i CNC.

These formats are incompatible; part programs for one type of controller cannot be used on the other types.

Press **F1** to select **Compustep** part program format.

Press **F2** to select **QUICKMILL/PC** or **QUICKDRILL/PC** format.

Press **F3** to select **QUICKMILL/Fanuc 15i** format.

Press **F4** to select Intimidator/21i format.

Press **F5** to select the bridge-style Eliminator or Quickdrill (B-Series).

### Part Program Filename

After specifying the format for the part program, **QUICKCODE®** will ask for a filename for the part program. The default filename will be the CAD filename with the .DXF or .CDL stripped off. If this default is appropriate for the part program, press **<enter>**. If a different filename is desired for the part program, backspace over the default and type in the correct filename.

part programs are stored on disk in the directory specified by variables in the configuration file. The variables are:

- **DATA\_DIR** for **Compustep** part programs
- **QM\_DATA\_DIR** for **QUICKMILL/PC** or **QUICKDRILL/PC** part programs
- **15I\_DATA\_DIR** for **QUICKMILL/15i** part programs
- **21I\_DATA\_DIR** for Intimidator/21i part programs
- **B-SERIES\_DATA\_DIR** for bridge-style Eliminator or Quickdrill part programs

For more information on these configuration variables, see **CHAPTER 5 - QUICKCODE® Configuration**.

NOTE: If a part program with the same name already exists, **QUIKCODE®** will prompt the operator to overwrite the old part program.

## **F1 Compustep part programs**

After specifying the part program filename, the part program will be written to disk. A counter will track the current line and the total number of lines to be written.

After writing out the part program, **QUIKCODE®** returns to the Tool Screen.

## **F2 QUICKDRILL/QUICKMILL part programs**

## **F3 Fanuc 15i part programs**

## **F4 Fanuc 21i part programs**

These three output formats are more complicated (and more flexible) than the Compustep format. The part program is assembled from “include” files and the operator is prompted for additional information.

Once the part program filename is entered **QUIKCODE®** will begin assembling the part program. This is a three step process, consisting of:

- Step 1: Inserting Include Files
- Step 2: Adding N Blocks (line numbers)
- Step 3: Specifying Machine Data

The first two steps are automatic and require no intervention on the part of the operator. The third step requires the operator to insert additional data.

### **Step 1: Inserting Include Files**

The first step in assembling the part program is to insert various “include” files. The “include” files are simply small program fragments that are common to most part programs. By changing the include files, the user can tailor **QUIKCODE®** to generate part programs in the correct format. For more information on the include files, see *CHAPTER 6 - Include Files*.

There are three include files:

**PRE\_PROG.INC** (for **QUICKMILL/PC** or **QUICKDRILL/PC**)

**15I\_PRE.INC** (for **QUICKMILL/15i**)

**21I\_PRE.INC** (for **Intimidator/21i**)

**B18\_PRE.INC** (for **B-Series**)

The “pre-program” include file (**PRE\_PROG.INC**, **15I\_PRE.INC**, **21I\_PRE.INC**, or **B18\_PRE.INC**) is inserted at the start of the part program, immediately following the **G20** or **G21** (inch or metric) code. The purpose of the “pre-program” include file is to start the part program in a known state. This is often referred to as a “safety block”. A typical file looks like:

```
1:      G80                      (CANCEL CANNED CYCLES)
```

```

2:  G28 X0 Y0 Z0 H0      (CANCEL ALL OFFSETS)
3:  T000                  (SEND 0 TO ATC)

```

The lines numbers (1:, 2:, etc.) are for illustration only and are not part of the include file. This example is for a *QUICKMILL/PC* or *QUICKDRILL/PC* controller. The corresponding *QUICKMILL/15i* include file (**15I\_PRE.INC**), Intimidator/21i (**21I\_PRE.INC**) or B-Series (**B18\_PRE.INC**) will be similar.

Line 1: inserts a **G80** code to cancel all canned cycles. It is good practice to always cancel canned cycles at the start of a part program.

Line 2: cancels the part offset on the **X**, **Y**, and **Z** axis and cancels the tool length offset (via the **H0** code).

Line 3: tells the CNC that the current tool number is **0**. This is normally only required for machines equipped with a carousel-style ATC (automatic tool changer). The procedure for using the carousel-style ATC is to always start and end the part program with no tool in the spindle. If your machine does not have a carousel-style ATC, this line can be removed from the include file. Refer to *CHAPTER 6 - Include Files* for information on changing the include files.

**TC.INC** (for *QUICKMILL/PC* or *QUICKDRILL/PC*)

**15I\_TC.INC** (for *QUICKMILL/15i*)

**21I\_TC.INC** (for Intimidator/21i)

**B18\_TC.INC** (for B-Series)

The “tool change” include file (**TC.INC**, **15I\_TC.INC**, **21I\_TC.INC**, or **B18\_TC.INC**) is added to the part program before the first tool is used and whenever a tool is changed thereafter. A typical tool change include file might look like:

```

1:  G80                  (CANCEL CANNED CYCLE)
2:  M05 M09             (SPINDLE OFF, COOLANT OFF)
3:  G28 X0 Y0 Z0 H0      (CANCEL ALL OFFSETS)
4:  G00 Z0              (RAISE Z TO CLEAR)
5:  ?G00 X5 Y5          (MOVE XY TO T/C POSITION)
6:  T### H###           (USE TOOL, LENGTH OFFSET)
7:  M06                 (TOOL CHANGE)

```

The lines numbers (**1:**, **2:**, etc.) are for illustration only and are not part of the include file. This example is for a *QUICKMILL/PC* or *QUICKDRILL/PC* controller.

Line 1: inserts a **G80** code to cancel all canned cycles. It is good practice to cancel canned cycles before a tool change.

Line 2: turns the spindle and coolant off. It is essential to stop the spindle and coolant pump before commanding a tool change.

Line 3: cancels the part offset on the **X**, **Y**, and **Z** axis and cancels the tool length offset (via the **H0** code). Again, this is simple good practice.

Line 4: raises the **Z** axis to its highest point from the table. This ensures the tool will clear the workpiece on the way to doing the tool change.

Line 5: rapid move to **X=5.0, Y=5.0**, to provide the operator easy access to the spindle for changing the tool. This move would not be required for machines fitted with an automatic tool changer. This line is optional, due to the “?” character in the first character position. When **QUICKCODE®** encounters a “?” character at the beginning of an include file, it will ask the user if the line should be included in the part program.

Line 6: inserts the codes **T###** and **H###** into the part program. These codes will cause **QUICKCODE®** to prompt for the **T** and **H** codes during final part program preparation.

Line 7: commands a tool change.

**END\_PROG.INC** (for **QUICKMILL/PC** or **QUICKDRILL/PC**)

**15I\_TC.INC** (for **QUICKMILL/15i**)

**21I\_TC.INC** (for Intimidator/21i)

**B18\_TC.INC** (for B-Series)

The “end of program” include file (**END\_PROG.INC**, **15I\_END.INC**, **21I\_END.INC**, or **B18\_END.INC**) is added to the part program at the end. The purpose of the “end of program” lines is to properly stop the part program (e.g, stop the spindle, stop the coolant, return the tool to the tool changer, etc.) A typical **END\_PROG.INC** might be:

```
1:  G80                      (CANCEL CANNED CYCLES)
2:  M05 M09                  (SPINDLE OFF, COOLANT OFF)
3:  G28 X0 Y0 Z0             (CANCEL ALL OFFSETS)
4:  T000 H000                (CANCEL TOOL OFFSETS)
5:  M06
6:  M02                      (END OF PROGRAM)
7:  %
```

The lines numbers (**1:**, **2:**, etc.) are for illustration only and are not part of the include file. This example is for a **QUICKMILL/PC** or **QUICKDRILL/PC** controller. The corresponding end-of-program include file for other CNC types will be similar.

Line 1: inserts a **G80** code to cancel all canned cycles. It is good practice to cancel canned cycles at the end of a part program.

Line 2: turns the spindle and coolant off. It is good practice to stop the spindle and coolant pump at the end of the part program.

Line 3: cancels the part offset on the **X**, **Y**, and **Z** axis.

Line 4: cancels the tool number and tool length offset.

Line 5: commands a tool change. This would result in ending the part program with no tool in the spindle. This is desirable on machines with an ATC, however, it may not be optimal for those without. Change the **END\_PROG.INC** file as required for your machine.

Line 6: commands an End of Program. It is important to have an end of program command in every part program. An alternative to the **M02** code is **M30, End of Program and Rewind**. The **M30** code is useful when the part program is repeated many times.

## Step 2: Adding N Blocks

The second step in assembling the part program is to add line number (**N** blocks) to each line. No operator intervention is required for this step.

## Step 3: Specifying Machine Data

**QUIKCODE®** will now begin listing the part program on the screen and will pause to “fill in the blanks” when it comes across codes that are not yet defined. The exact order of the following steps depends on how the include files are written. A typical sequence of prompts follows.

### Tool Number (T code)

When a line with a **T###** code is encountered, **QUIKCODE®** prompts the user for the tool number for this tool, as in the following example:

```
%
{Program: TUBESHT1 Created: 08-14-1996 - 13:30:23 by: QUIKCODE v3.10}
{
N0040 G20 (SELECT INCH UNITS)
N0060 G80 (CANCEL CANNED CYCLES)
N0080 G28 X0 Y0 Z0 H0 (CANCEL ALL OFFSETS)
N0100 T000 (SEND 0 TO ATC)
N0120 G80 (CANCEL CANNED CYCLE)
N0140 M05 M09 (SPINDLE OFF, COOLANT OFF))
N0160 G28 X0 Y0 Z0 H0 (CANCEL ALL OFFSETS))
N0180 G00 Z0 (RAISE Z TO CLEAR))
N0200 G00 X5 Y5 (MOVE XY TO T/C POSITION))
N0220 T### H### (USE TOOL, LENGTH OFFSET)

Enter tool number for this T code: |||||
```

**Figure 21**

In this example, a **T###** code was detected in line **N0220**. **QUIKCODE®** is asking the user what tool number should be inserted in place of **T###**. Enter the required tool number and press **<enter>**. It is not necessary to enter the letter “**T**”, just the number itself.

Once the first tool number has been entered, **QUIKCODE®** will automatically default to the next tool number when the next **T###** is encountered.

### Tool Length Offset Number (H code)

When a line with a **H###** code is encountered, **QUIKCODE®** prompts for the tool length offset to use. Normally this number is the same as the tool number, so **QUIKCODE®** will offer the last tool number entered as a default. If this is acceptable, simply press **<enter>**. If not, erase the default by pressing the **b** key and then type in a new tool length offset number.

### Tool Description

The user can add a description of the tool in the comment field at the end of the line. If no description is desired, just press **<enter>**.

Note: It is good practice to use the tool description field to describe the tool in plain text. The machine operator can read the description as the part program is run, eliminating confusion as to which tool is required.

Once the tool data has been entered, **QUIKCODE**® will ask if it is OK to commit this line to the part program. The screen will look like:

```
%
{Program: TUBESHT1 Created: 08-14-1996 - 13:30:23 by: QUIKCODE v3.10}
N0040 G20 (SELECT INCH UNITS)
N0060 G80 (CANCEL CANNED CYCLES)
N0080 G28 X0 Y0 Z0 H0 (CANCEL ALL OFFSETS)
N0100 T000 (SEND 0 TO ATC)
N0120 G80 (CANCEL CANNED CYCLE)
N0140 M05 M09 (SPINDLE OFF, COOLANT OFF))
N0160 G28 X0 Y0 Z0 H0 (CANCEL ALL OFFSETS))
N0180 G00 Z0 (RAISE Z TO CLEAR))
N0200 G00 X5 Y5 (MOVE XY TO T/C POSITION))
N0220 T001 H001 (USE TOOL, LENGTH OFFSET) (1.25" CARBIDE INSERT DRI

Commit this line to the output file? [Y/N] Y_
```

**Figure 22**

If the line of code is correct, press **<enter>** to commit (or write) the line to the part program. If there are mistakes in the line, enter **N** to re-enter the values for the tool.

#### **Part Offset (G54 P code)**

When a **G54 P###** code is encountered, **QUIKCODE**® will prompt the user to enter the part offset for this tool. This value is used by the **QUICKMILL**® to retrieve the proper part offset from the Part Offset Table on the **QUICKMILL**.

**QUIKCODE**® will then prompt the user to confirm this line is correct before committing the line to the part program. Press **Y** and press **<enter>** to tell **QUIKCODE**® to write the line, otherwise press **N** then **<enter>** to re-enter the part offset number.

If the output format is intended for a Fanuc CNC, (e.g., **QUICKMILL/15i**, **Intimidator/21i**, **B-Series**), **QUIKCODE**® will prompt for **G54 - G59** instead of a **P** value.

#### **Spindle Speed (RPM) (S code)**

When **QUIKCODE**® encounters an **S????** code it will prompt the user for the spindle speed for the current tool, as in this example:

```

N0160 G28 X0 Y0 Z0 H0 (CANCEL ALL OFFSETS))
N0180 G00 Z0 (RAISE Z TO CLEAR))
N0200 G00 X5 Y5 (MOVE XY TO T/C POSITION))
N0220 T001 H001 (USE TOOL, LENGTH OFFSET) (1.25" CARBIDE DRILL)
N0240 M06 (TOOL CHANGE)
N0260 G54 P001
N0280 G00 X-003.0000 Y-012.1250
N0300 S7777 M03 (SET SPINDLE SPEED, SPINDLE ON FORWARD)

```

Enter spindle RPM for this tool: |||||

**Figure 23**

Enter the spindle RPM and press **<enter>**. **QUICKCODE®** will then prompt the user to confirm this value before writing the line to the part program.

### Coolant Nozzle Type (M code)

When **QUICKCODE®** encounters an **M?? M08** code, it will prompt the user for the coolant type to use for the current tool. There are three choices, as shown below:

```

N0220 T001 H001 (USE TOOL, LENGTH OFFSET) (1.25" CARBIDE DRILL)
N0240 M06 (TOOL CHANGE)
N0260 G54 P001
N0280 G00 X-003.0000 Y-012.1250
N0300 S1500 M03 (SET SPINDLE SPEED, SPINDLE ON FORWARD)
N0320 M?? M08 (SELECT NOZZLE, COOLANT ON)

```

Flood, Thru-tool, or No coolant nozzle [F, T, or N]: |||

**Figure 24**

Enter **F** for flood coolant, **T** for thru-tool coolant, and **N** for no coolant. The appropriate codes will be inserted in the part program.

### Feed Rate (F code)

An **F????** code will trigger **QUICKCODE®** to prompt the user for the feed rate for the current tool. Enter the desired feed rate, in units per minute, when prompted.

### Canned Drill Cycle (G code)

For all formats other than Compustep 12-bit, a **G** code determines the type of drill cycle to use. If a canned drill cycle has already been specified (for a previous tool), it will be the default. If the default is acceptable, just press **<enter>**. If the default is not acceptable, use the **y** and **v** cursor keys to select the desired canned drill cycle. **QUICKCODE®** will then ask for any extra parameters specific to the selected canned drill cycle, such as dwell time, fine bore shift distance, peck dwell time, etc.

Refer to the following tables for a list of the canned cycles included with each type of CNC:

Table 1 on page 43 lists the canned drill cycles available on a **QUICKMILL/PC** and **QUICKDRILL/PC** CNC.

Table 2 on page 44 lists the canned drill cycles available on a **QUICKMILL/15i** CNC.

Table 3 on page 45 lists the canned drill cycles available on the **Intimidator/21i** CNC or a **B-Series**.

Depending on which drill cycle is chosen, **QUICKCODE®** will prompt the user for additional information, as required for that drill cycle.

## G89 - Auto-Circle Canned Cycle

The G89 - Auto-Circle Canned Cycle allows the part programmer to describe a complex circular operation with one command. Starting from a center point, this cycle will automatically interpolate the arc-in and arc-out moves to ensure the resulting surface is cut smoothly. The same circle can be cut at up to 5 different Z axis depths, allowing for circular pocketing and the cutting of multiple grooves inside an existing hole. Since the G89 command is modal, the part program size and complexity can be drastically reduced.

The G89 code will only run on the *QUICKMILL/PC* and *QUICKDRILL/PC* that have this option purchased.

If the **G89** - Auto-Circle Canned Cycle is chosen, **QUICKCODE**® will prompt for the various data that this cycle requires. For a detailed discussion of the **G89** - Auto-Circle Canned Cycle, see the **G89** - Auto-Circle Canned Cycle option manual, *QUICKMILL* Part #09100023.

The sequence of prompts for a **G89** canned cycle are as follows:

Enter 1st depth (G89 Z word):

Enter the depth for the first depth of cut for the **G89** cycle. If a blank entry is given, **QUICKCODE**® will stop prompting for **G89** data.

Enter diameter of the circle (G89 D word):

Enter the diameter of the circle for the **G89** cycle. The meaning of this word depends on whether the tool table diameter will be used (if a **P** word is used) or not. If a **P** word is used later, this **D** value will be interpreted as the diameter of the resulting cut circle. If no **P** word is used, this **D** value is the diameter of the tool centerline.

Use **y** and **v** to select circle direction: Clockwise

Use the **y** and **v** cursor keys to select the direction of cut for the **G89** canned cycle. Press **<enter>** when the desired direction is selected.

Enter number of revolutions (G89 L word):

Enter either **1** or **2** to set the number of revolutions of the circle for the **G89** canned cycle.

How much stock to leave on radius on 1st pass:

If the number of revolutions is **2** (see the previous prompt for the **L** word), **QUICKCODE**® will prompt for a “leave stock” value. This is the amount to hold off on the diameter on the first pass around the circle.

Enter the tool table entry for the tool diameter (P word):

If a value is entered here, the **G89** cycle will use this entry’s diameter value from the tool table on the CNC. This allows the part programmer to specify the true diameter of the resulting cut circle, leaving the CNC to calculate the tool path centerline.

If no value is entered for the **P** word, the diameter (**D** word) will be interpreted as the tool path centerline.

Enter 2nd depth (G89 A word):

If the same circle is to be cut at a second depth, enter it now as the **A** word. The CNC will repeat the same circle at this depth.

If a blank entry is given at this prompt, the remaining 3 prompts (for the **B**, **I**, and **J** words) will be skipped.

Enter 3rd depth (G89 B word):

If the same circle is to be cut at a third depth, enter it now as the **B** word. The CNC will repeat the same circle at this depth.

If a blank entry is given at this prompt, the remaining 2 prompts (for the **I** and **J** words) will be skipped.

Enter 4th depth (G89 I word):

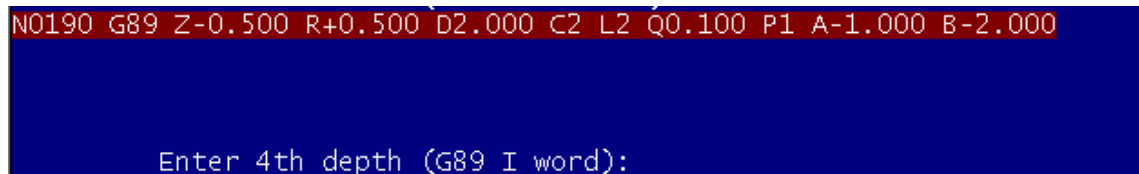
If the same circle is to be cut at a fourth depth, enter it now as the **I** word. The CNC will repeat the same circle at this depth.

If a blank entry is given at this prompt, the remaining prompt (for the **J** word) will be skipped.

Enter 5th depth (G89 J word):

If the same circle is to be cut at a fifth depth, enter it now as the **J** word. The CNC will repeat the same circle at this depth.

See Figure 25 for an example of the **G89** - Auto-Circle Canned Cycle data being entered.



**Figure 25 Entering G89 - Auto-Canned Circle data.**

Due to width limitations on the screen, the **G89** line may wrap at 80 columns. The complete line will be written to the part program regardless of its length.

### Layered Drill Cycle Codes

After entering the basic drill cycle information, **QUIKCODE**® may offer to include layered drill cycle codes. Whether **QUIKCODE**® offers this is determined by the following three conditions:

1. The **OFFER\_M41** variable must be set to **-1** in the **QUIKCODE**® configuration file. See **OFFER\_M41** on page 56 for details on setting this variable.
2. The type of CNC must be one that supports the layered drill cycle. Currently this is the **QUICKMILL/PC**, **QUICKDRILL/PC**, **QUICKMILL/15i**, and **Intimidator/21i**.
3. The drill cycle must be one of these supported by the current CNC type, as follows:

**QUICKMILL/PC** or **QUICKDRILL/PC**      **G73, G76, G77, G78, G79, G81, G82, G83, G86, and G88**

**QUICKMILL/15i**      **G81**

**Intimidator/21i**      **G81**

If all these conditions are true, then **QUICKCODE®** will prompt:

Add M41 codes to this drill cycle? [Y or N]:

Answering **Y** to this prompt will cause **QUICKCODE®** to start prompting for the layered drill cycle data according to the capabilities of the CNC type:

**QUICKMILL/PC** or **QUICKMILL/PC**:

Enter the 1st depth:

This is the depth at which the following feedrate, spindle RPM and peck multiplier will be applied. The term “1st” will change to “2nd”, “3rd”, etc. for each depth.

Entering a blank value (not zero, which is a valid depth) at this prompt will end the entry of layered drill cycle values.

Enter the feedrate percentage at this depth:

Enter the feedrate override as a percentage of the programmed feed rate (the **F** word). This value must be in the range of the CNC’s acceptable feedrate override, currently 10% to 200%.

Enter the spindle speed percentage at this depth:

Enter the spindle override as a percentage of the programmed spindle RPM (the **S** word). This value must be in the range of the CNC’s acceptable spindle RPM override, currently 10% to 300%.

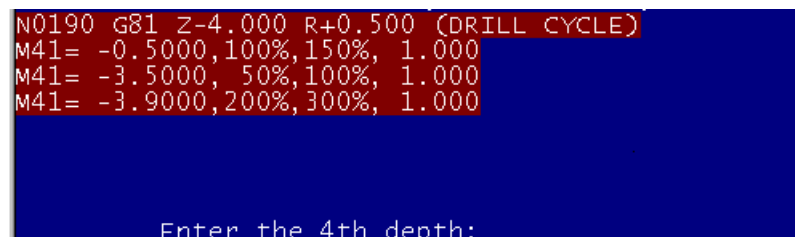
Enter the peck multiplier at this depth:

The peck multiplier is a value that is multiplied by the peck interval (the **Q** word) at this depth.

This sequence of prompts will repeat until:

- a total of 8 depths is entered
- or:
- no value is entered for the depth

As each entry is made, the values will be written on the screen below the drill cycle line, as in Figure 26:



```
N0190 G81 Z-4.000 R+0.500 (DRILL CYCLE)
M41= -0.5000,100%,150%, 1.000
M41= -3.5000, 50%,100%, 1.000
M41= -3.9000,200%,300%, 1.000
Enter the 4th depth:
```

**Figure 26 - M41 Codes in Entry Order**

Once all entries are done, **QUICKCODE®** will re-order the lines into the correct order, since **M41** codes must come before the drill cycle line. Line numbers will also be added to each line. See Figure 27 for an example.

```

N0187 M41=-0.500,100%,150%,1.000
N0188 M41=-3.500,50%,100%,1.000
N0189 M41=-3.900,200%,300%,1.000
N0190 G81 Z-4.000 R+0.500 (DRILL CYCLE)

Commit this line to the output file? [Y/N] Y_

```

**Figure 27 -Re-ordered and Number M41 Lines**

For detailed information and examples of the layered drill cycle on the *QUICKMILL/PC* and *QUICKDRILL/PC*, see the corresponding Operator's Manual.

*QUICKMILL/15i* and *Intimidator/21i*:

Enter the 1st depth (A value):

This is the depth at which the following feedrate and spindle RPM will be applied. The term "1st" will change to "2nd", "3rd", etc. for each depth. Likewise, the **A** will change to indicate the specific CNC word that is being entered (**A**, **I**, **U**, **D**, or **M**).

Entering a blank value (not zero, which is a valid depth) at this prompt will end the entry of layered drill cycle values.

Enter the feedrate at this depth (B value):

Enter the feedrate at this depth. This value must be in the range of the CNC's acceptable feedrate for the Z axis. The label **B** will change for each depth to indicate the specific CNC word that is being entered (**B**, **J**, **V**, **E**, or **Q**)

Enter the spindle RPM at this depth (C value):

Enter the spindle RPM at this depth. This value must be in the range of the CNC's acceptable spindle RPM values. The label **C** will change for each depth to indicate the specific CNC word that is being entered (**C**, **K**, **W**, **F**, or **R**)

This sequence of prompts will repeat until:

- a total of 5 depths is entered.

or:

- no value is entered for the depth.

As each entry is made, the values will be written on the screen on the drill cycle line, as in Figure 28. Note that due to width limitations, the line will wrap on the screen if more than two layers are entered.

```

N0150 G66 P9041 Z-4.0000 R+0.5000 A-0.5 B15.5 C1000 I-2. J10. (LAYERED DRILL CYC
LE)

Enter the spindle RPM at this depth (K value): 2000

```

**Figure 28 - Layered Drill Cycle Codes for /15i or /21i**

Once all entries are done, *QUICKCODE*® will re-write the line on the screen. Due to width limitations, the line will be chopped off at 80 characters on screen. However, the complete line will be written to the part program file.

For detailed information and examples of the layered drill cycle on the *QUICKMILL/15i* CNC, see the *QUICKDRILL/15i* Option: Layered Drill Cycle, part #09500006.

For detailed information and examples of the layered drill cycle on the Intimidator/21i CNC, see the Intimidator/21i Option: Layered Drill Cycle, part #09400006.

### Hole Data

Once the canned drill cycle information is entered, **QUICKCODE**® will proceed to write the X & Y coordinates for all the holes in the pattern. If there are additional tools in the part program, the above sequence of prompts will be repeated to define operation for each tool.

### Copy to Floppy

Once the complete part program has been written, **QUICKCODE**® may ask if the part program should also be written to a floppy disk (for transferring to the CNC, or perhaps for backup purposes). The configuration file variable **OFFER\_FLOPPY\_COPY** determines whether **QUICKCODE**® offers to copy the just-created part program to the floppy drive. See page 56 for details on setting this variable and the related **FLOPPY\_COPY\_DRIVE** variable.

If **QUICKCODE**® is set to offer copying to floppy disk, the following prompt will be given after the part program is written to the normal output directory



If **Y** is pressed, **QUICKCODE**® will attempt to write the part program to the floppy drive specified by the **FLOPPY\_COPY\_DRIVE** configuration variable. Before pressing **Y**, ensure that:

- a floppy disk is in the specified floppy drive;
- the floppy disk has sufficient free space;
- the floppy disk is no write-protected;

**QUICKCODE**® will verify the above conditions before it attempts to write the file to the floppy disk. A message will be given as to the success or failure of writing the file.

| <b>Table 1 - QUICKDRILL/PC &amp; QUICKMILL/PC Canned Drill Cycles</b> |                                                                                                                                                                                                                                                               |                                    |                                 |
|-----------------------------------------------------------------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|------------------------------------|---------------------------------|
| <b>Code</b>                                                           | <b>Description</b>                                                                                                                                                                                                                                            | <b>Format</b>                      | <b>Example</b>                  |
| <b>G73</b>                                                            | Peck Drill Cycle. Feeds in distance Q, jumps back distance J (optional), dwells for time D (optional), feeds another distance Q. Repeats until Z value is reached. Then rapid to R value.                                                                     | G73 Z#.## R#.## Q##### J#.## D#.## | G73 Z-1.125 R0.5 Q.125 J.05 D.1 |
| <b>G77</b>                                                            | Peck Drill Cycle with Bottom Stop. Same as G73, but spindle is shut off when Z value is reached.                                                                                                                                                              | see G73                            | see G73                         |
| <b>G78</b>                                                            | Peck Drill Cycle with Bottom Dwell. Same as G73, but adds a dwell for P seconds before rapid to R value.                                                                                                                                                      | See G73, add P#.##                 | see G73                         |
| <b>G79</b>                                                            | Peck Drill Cycle with Bottom Stop and Bottom Dwell. Combines G78 and G79                                                                                                                                                                                      | see G77, G78, G73                  | see G77, G78, G73               |
| <b>G83</b>                                                            | Peck Drill Cycle with Full Extract. Similar to G73, with Z axis retracted to R value after each Q value.                                                                                                                                                      | See G73                            | See G73                         |
| <b>G76</b>                                                            | Fine Bore Cycle. Feed to Z value at current feed rate. Stop spindle. Orient Spindle. Move Y axis shift distance (Q value) in negative direction. Retract Z axis to R value. G76 is only available on QUICKMILLS equipped with the Spindle Orientation option. | G76 Z#.## R#.## Q#.##              | G76 Z-2.125 R0.100 Q0.025       |
| <b>G81</b>                                                            | Simple canned drill cycle. Feed to Z value at current feed rate, rapid to R value                                                                                                                                                                             | G81 Z#.## R#.##                    | G81 Z-2.50 R+0.10               |
| <b>G82</b>                                                            | Canned Drill Cycle with Dwell. Same as G81, adds a dwell for P seconds before retracting to R value                                                                                                                                                           | G82 Z#.## R#.## P#.##              | G82Z-1.75 R0 P1.0               |
| <b>G86</b>                                                            | Canned Drill Cycle with Bottom Stop. Same as G81, but stops the spindle rotation at the bottom of the hole (at the Z value) before retracting to R value.                                                                                                     | G81                                | G81                             |
| <b>G88</b>                                                            | Canned Drill Cycle with Dwell and Bottom Stop. Combines G82 and G86.                                                                                                                                                                                          | see G86, G82, G81                  | see G86, G82, G81               |
| <b>G74</b>                                                            | Tap Cycle, Left Hand Tap. Spindle on reverse, feeds tap to Z value at current feed rate, stop and then forward spindle, feed tap to Z = 0 at current feed rate, then rapid to R value.                                                                        | G74Z#.## R#.##                     | G74Z-2.25 R.5                   |
| <b>G84</b>                                                            | Tap Cycle, Right Hand Tap. Spindle on forward, feeds tap to Z value at current feed rate, stop and then reverse spindle, feed tap to Z= 0 at current feed rate, then rapid to R value.                                                                        | G84Z#.## R#.##                     | G84 Z-3.10 R1.00                |

**Table 2 - QUICKMILL/15i Canned Drill Cycles**

| Code | Description                                                                                                                                                                                             | Format                      | Example                     |
|------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|-----------------------------|-----------------------------|
| G73  | Peck Drill Cycle. Feeds in distance Q, jumps back distance specified in parameter #6210, repeats until Z value is reached. Then rapid to R value or initial level.                                      | G73 Z#. # R#. # Q#. # F#. # | G73 Z-1.125 R0.5 Q.125 F25. |
| G74  | Tap Cycle, Left Hand Tap. Spindle on reverse, feeds tap to Z value at current feed rate, stop and then forward spindle, feed tap to Z = 0 at current feed rate, then rapid to R value or initial level. | G74Z#. # R#. # F#. #        | G74Z-2.25 R.5 F15.          |
| G76  | Fine Bore Cycle. Feed to Z value at current feed rate. Stop spindle. Orient Spindle. Move Y axis shift distance (Q value) in negative direction. Retract Z axis to R value or initial level.            | G76 Z#. # R#. # Q#. # F#. # | G76 Z-2.0 R0.1 Q0.025 F10.  |
| G81  | Canned drill cycle. Feed to Z value at current feed rate, rapid to R value or initial level.                                                                                                            | G81 Z#. # R#. # F#. #       | G81 Z-2.50 R+0.10 F25.0     |
| G82  | Drill Cycle with Bottom Dwell. Same as G81, but adds a dwell for P milliseconds before rapid to R value or initial level.                                                                               | G82 Z#. # R#. # P## F#. #   | G82 Z-2.50 R+0.10 P500 F10. |
| G83  | Peck Drill Cycle with Full Extract. Similar to G73, with Z axis retracted to R value after each Q value.                                                                                                | See G73                     | See G73                     |
| G84  | Tap Cycle, Right Hand Tap. Spindle on forward, feeds tap to Z value at current feed rate, stop and then reverse spindle, feed tap to Z= 0 at current feed rate, then rapid to R value or initial level. | G84 Z#. # R#. # F#. #       | G84 Z-3.10 R1.00 F10.       |
| G85  | Boring cycle. Feed to Z value at current feed rate, feed to R value or initial level.                                                                                                                   | G85 Z#. # R#. # F#. #       | G85 Z-2.50 R+0.10 F25.0     |
| G86  | Boring cycle with bottom stop. Feed to Z value at current feed rate, stop spindle, feed to R value or initial level.                                                                                    | G86 Z#. # R#. # F#. #       | G86 Z-2.50 R+0.10 F25.0     |
| G87  | Boring cycle with bottom shift and manual retract. Feed to Z value at current feed rate, stop spindle, shift Y by Q amount, allow operator to manually retract tool..                                   | G87 Z#. # R#. # Q#. # F#. # | G87 Z-2.5 R+0.1 Q0.05 F5.0  |
| G88  | Boring cycle with bottom dwell and manual retract. Feed to Z value at current feed rate, dwell P milliseconds, stop spindle, allow operator to manually retract tool.                                   | G88 Z#. # R#. # P## F#. #   | G88 Z-2.5 R+0.1 P500 F25.0  |
| G89  | Boring cycle with bottom dwell. Feed to Z value at current feed rate, dwell P milliseconds, feed to R value or initial level.                                                                           | G89 Z#. # R#. # P## F#. #   | G89 Z-2.5 R+0.1 P750 F5.0   |

**Table 3 - Intimidator/21i or B-Series Canned Drill Cycles**

| Code | Description                                                                                                                                                                                             | Format              | Example                     |
|------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|---------------------|-----------------------------|
| G73  | Peck Drill Cycle. Feeds in distance Q, jumps back distance specified in parameter #6210, repeats until Z value is reached. Then rapid to R value or initial level.                                      | G73 Z## R## Q## F## | G73 Z-1.125 R0.5 Q.125 F25. |
| G74  | Tap Cycle, Left Hand Tap. Spindle on reverse, feeds tap to Z value at current feed rate, stop and then forward spindle, feed tap to Z = 0 at current feed rate, then rapid to R value or initial level. | G74Z## R## F##      | G74Z-2.25 R.5 F15.          |
| G76  | Fine Bore Cycle. Feed to Z value at current feed rate. Stop spindle. Orient Spindle. Move Y axis shift distance (Q value) in negative direction. Retract Z axis to R value or initial level.            | G76 Z## R## Q## F## | G76 Z-2.0 R0.1 Q0.025 F10.  |
| G81  | Canned drill cycle. Feed to Z value at current feed rate, rapid to R value or initial level.                                                                                                            | G81 Z## R## F##     | G81 Z-2.50 R+0.10 F25.0     |
| G82  | Drill Cycle with Bottom Dwell. Same as G81, but adds a dwell for P milliseconds before rapid to R value or initial level.                                                                               | G82 Z## R## P## F## | G82 Z-2.50 R+0.10 P500 F10. |
| G83  | Peck Drill Cycle with Full Extract. Similar to G73, with Z axis retracted to R value after each Q value.                                                                                                | See G73             | See G73                     |
| G84  | Tap Cycle, Right Hand Tap. Spindle on forward, feeds tap to Z value at current feed rate, stop and then reverse spindle, feed tap to Z= 0 at current feed rate, then rapid to R value or initial level. | G84 Z## R## F##     | G84 Z-3.10 R1.00 F10.       |
| G85  | Boring cycle. Feed to Z value at current feed rate, feed to R value or initial level.                                                                                                                   | G85 Z## R## F##     | G85 Z-2.50 R+0.10 F25.0     |
| G86  | Boring cycle with bottom stop. Feed to Z value at current feed rate, stop spindle, feed to R value or initial level.                                                                                    | G86 Z## R## F##     | G86 Z-2.50 R+0.10 F25.0     |
| G87  | Boring cycle with bottom shift and manual retract. Feed to Z value at current feed rate, stop spindle, shift Y by Q amount, allow operator to manually retract tool..                                   | G87 Z## R## Q## F## | G87 Z-2.5 R+0.1 Q0.05 F5.0  |
| G88  | Boring cycle with bottom dwell and manual retract. Feed to Z value at current feed rate, dwell P milliseconds, stop spindle, allow operator to manually retract tool.                                   | G88 Z## R## P## F## | G88 Z-2.5 R+0.1 P500 F25.0  |
| G89  | Boring cycle with bottom dwell. Feed to Z value at current feed rate, dwell P milliseconds, feed to R value or initial level.                                                                           | G89 Z## R## P## F## | G89 Z-2.5 R+0.1 P750 F5.0   |



# CHAPTER 4

## COPY FILES TO FLOPPY DISK

---

To transfer part programs to the *QUICKMILL*/PC or *QUICKDRILL*/PC, use the DOS **COPY** command to copy the part programs to a floppy disk.

In the following examples, assume that:

- the **QUICKCODE**® software is installed in **C:\QC4**
- the *QUICKDRILL*/PC part programs are stored in **C:\QC4\PARTS**
- the *QUICKMILL* FANUC control part programs are stored in **C:\QC4\PROGRAMS**
- the floppy drive is **A:**

First, change to the directory that holds the part programs you wish to copy. Assuming *QUICKMILL*/PC part programs, type:

```
C: <enter>
```

```
CD \QC4\QUICKMILL <enter>
```

Depending on how your computer is configured, you may see the DOS prompt change to:

```
C:\QC4\QUICKMILL>
```

Use the **COPY** command to copy the desired files. If the part program **95-1450.DRL** is to be copied to the **A:** floppy disk, the command is:

```
COPY 95-1450.DRL A: /V <enter>
```

The **/V** switch tells DOS to verify that the copy operation was successful.

To ensure that the part program was copied with no errors, use the **FC** (File Compare) command. For this example, type:

```
FC /B 95-1450.DRL A:
```

If there are no errors reported, then the part program on the floppy disk is exactly the same as the part program on the hard disk. If any errors or differences are reported, repeat the copy command. If errors or differences are reported again, you may want to reformat the floppy disk or use another floppy disk.

**Note!** It is very important that the files be compared before allowing them to be copied onto the *QUICKMILL*/PC or *QUICKMILL*/PC. Attempting to run part programs that are incomplete or corrupted may result in incorrect parts being produced at the machine.

### Copying Multiple Files

The DOS **COPY** command can copy multiple files that have similar file names. For instance, the files:

```
95-1450.DRL
```

```
95-1450.TAP
95-1450.CSK
95-1450.SPT
```

These four files could be copied with one command:

```
COPY 95-1450.* A: <enter>
```

The “\*” character is a *wildcard* that matches any characters. In other words, this command says:

Copy the files that begin with “95-1450”, then a period “.” followed by *anything*.

Of course, after copying the files, you should compare them with the **FC** command. Again, the wildcard character can be used effectively:

```
FC /B 95-1450.* A: <enter>
```

There are many powerful ways to use the DOS *wildcard* characters with the **COPY** command. Consult your DOS manual for details.

# CHAPTER 5

## CONFIGURATION

---

The following is a description of the various configuration parameters contained in the **QUIKCODE**® Configuration File. This configuration file is essential to the operation of **QUIKCODE**®. It should only be modified by, or under the direction of, a **QUICKMILL** Inc. technician.

The **QUIKCODE**® configuration file is a simple ASCII text file located in the same directory as the **QUIKCODE.EXE** program. The configuration file is named **QUIKCODE.CFG**. A “simple ASCII text file” is a file that contains only text (numbers and letters), with no extra formatting characters. The configuration file can be edited with any ASCII text editor such as DOS **EDIT**, **EDLIN**, **Qedit**, etc. Do not use a word processor to change the configuration file. Word processors add special formatting characters to the file. These formatting characters will cause the configuration file to be unreadable.

The **QUIKCODE**® configuration file contains configuration parameters in the form:

PARAMETER NAME = PARAMETER VALUE

The parameter name and parameter value can be upper or lower case.

An example:

DATA\_DIR = C:\QC4\PARTS\

**DATA\_DIR** is the name of the parameter, and **C:\QC4\PARTS\** is the value assigned to it. The **QUIKCODE**® software uses the parameter **DATA\_DIR** to determine where to store **Compustep**® part programs.

Lines beginning with a : (colon) are comment lines and are ignored. Comment lines are used to document the configuration file.

The **QUIKCODE**® configuration file is only read when the **QUIKCODE**® software starts up. For a change in the configuration to be active, **QUIKCODE**® must be re-started.

### Configuration Parameters

#### 15I\_DATA\_DIR

The **15I\_DATA\_DIR** variable tells **QUIKCODE**® where to store part programs created for a **QUICKMILL/15i** CNC. This directory should not be the same as the directory for other formats.

The value of the **15I\_DATA\_DIR** variable should be in the form of:

d:\directory\

where:

**d:** is a valid drive letter

**\directory\** is the directory on the drive to store files in. Note that the directory must have a trailing backslash (“\”).

An example of a **15I\_DATA\_DIR** parameter is:

```
15I_DATA_DIR=C:\QC4\FANUC\
```

When the **QUIKCODE**® software is installed from the installation floppy disk, the **15I\_DATA\_DIR** is set to:

```
d:\install_dir\fanuc15i\
```

where:

**d:\install\_dir** is the directory **QUIKCODE**® is installed in.

## **21I\_DATA\_DIR**

The **21I\_DATA\_DIR** variable tells **QUIKCODE**® where to store part programs created for an Intimidator/21i CNC. This directory should not be the same as the directory for other formats.

The value of the **21I\_DATA\_DIR** variable should be in the form of:

```
d:\directory\
```

where:

**d:** is a valid drive letter

**\directory\** is the directory on the drive to store files in. Note that the directory must have a trailing backslash ("\").

An example of a **21I\_DATA\_DIR** parameter is:

```
21I_DATA_DIR=C:\QC4\FANUC\
```

When the **QUIKCODE**® software is installed from the installation floppy disk, the **21I\_DATA\_DIR** is set to:

```
d:\install_dir\fanuc21i\
```

where:

**d:\install\_dir** is the directory **QUIKCODE**® is installed in.

## **ADD\_HOLE\_NUMS**

Sets whether **QUIKCODE**® should output the hole number in a comment field on each line.

**ADD\_HOLE\_NUMS=-1** - **QUIKCODE**® will add hole numbers

**ADD\_HOLE\_NUMS=0** - **QUIKCODE**® will **NOT** add hole numbers

If holes numbers are added, each hole in the pattern will have a comment field appended to it, as in the following program fragment:

```
...  
M5 S300 (SPINDLE FWD)  
G81 Z-2.0 R0.5 F10. (DRILL CYCLE)  
X2.0 Y0.0 (#2)  
T000 H000 (#3)  
...etc...
```

Hole numbers can be used by the operator to quickly locate a re-start point, since comments fields are searchable on the **QUICKMILL**, **QUICKDRILL** and Fanuc 15i CNCs.

## B-SERIES\_DATA\_DIR

The **B-SERIES\_DATA\_DIR** variable tells **QUIKCODE**® where to store part programs created for a B-Series machine (bridge-style Eliminator or Quickdrill with a Fanuc 16i or 18i CNC). This directory should not be the same as the directory for other formats.

The value of the **B-SERIES\_DATA\_DIR** variable should be in the form of:

d:\directory\

where:

**d:** is a valid drive letter

**\directory\** is the directory on the drive to store files in. Note that the directory must have a trailing backslash ("").

An example of a **B-SERIES\_DATA\_DIR** parameter is:

B-SERIES\_DATA\_DIR=C:\QC4\B-SERIES\

When the **QUIKCODE**® software is installed from the installation floppy disk, the **B-SERIES\_DATA\_DIR** is set to:

d:\install\_dir\fanuc21i\

where:

**d:\install\_dir** is the directory **QUIKCODE**® is installed in.

## CDL\_DIR

The **CDL\_DIR** variable tells **QUIKCODE**® where the CAD CDL files are stored. When the operator processes a CDL file in **QUIKCODE**®, all files with the .CDL extension in the **CDL\_DIR** directory are listed.

The **CDL\_DIR** variable should be in the form of:

d:\directory\

where:

**d:** a valid drive letter

**\directory\** is the directory on the drive that CDL files are stored in. Note that the directory must have a trailing backslash ("").

An example of a **CDL\_DIR** parameter is:

CDL\_DIR=C:\QC4\CDL\

When the **QUIKCODE**® software is installed from the installation floppy disk, the **CDL\_DIR** is set to:

d:\install\_dir\CDL\

where:

**d:\install\_dir** is the directory **QUIKCODE**® is installed in.

It may be preferable to change this variable to point to the CAD software's standard CDL directory. For instance, CADKEY normally stores CDL files in the directory **C:\CADKEY\CDL**. To change **QUIKCODE**® to use the CADKEY CDL directory, change the **CDL\_DIR** variable to:

CDL\_DIR=C:\CADKEY\CDL\

A drawback to this method is that the **QUIKCODE**® operator will see all CDL files, even those that are not destined to be converted into part programs. It may be safer or easier to keep the CDL directory as it is installed and only write CDL files to that directory when they are to be converted to part programs.

## **COOLANT\_DEFAULT**

The **COOLANT\_DEFAULT** provides a method to have **QUIKCODE**® assume a default value for the type of coolant. When **QUIKCODE**® prompts for the type of coolant, the letter set here will be the default, allowing the user to simply press ENTER to use this type. The operator can, of course, change the coolant type if required.

**COOLANT\_DEFAULT=letter**

where: **letter** is a single letter from the following list:

- N** - no coolant
- F** - flood coolant
- T** - thru-tool coolant
- A** - air coolant
- M** - mist coolant

Note that not all types of coolant systems are available on all machines; commanding a non-existent coolant system may result in an alarm on the CNC, or it may result in the incorrect (or no) coolant for the tool.

The specific M-code output will be appropriate for the type of machine & CNC.

## **COORDINATE\_SYSTEM\_ROTATION\_DEFAULT**

The **COORDINATE\_SYSTEM\_ROTATION\_DEFAULT** variable tells **QUIKCODE**® what to offer as a default R word (rotation angle) for the **G68 - Coordinate System Rotation** command. Some examples

**COORDINATE\_SYSTEM\_ROTATION\_DEFAULT=#nnn**

Output a specific macro variable for the **R** word, where **nnn** is the macro variable number that stores the desired rotation angle. For instance, if the configuration file has the following entry:

**COORDINATE\_SYSTEM\_ROTATION\_DEFAULT=#139**

the **G68** command will have **R#139** included. When this line is executed the CNC will replace **#139** with the value in macro variable **#139**. Obviously, it is important that the correct angle be stored in macro variable **#139**, perhaps by a probe routine or operator input.

Any macro variable available on the CNC can be used, but it is imperative that the variable is free to use (not used for some other purpose) and that it be set to the correct value.

**COORDINATE\_SYSTEM\_ROTATION\_DEFAULT=###**

Outputs **R###** on the **G68** line. It will be necessary to edit the **R###** word before running the part program, as an **R###** word will cause an alarm on the CNC.

**COORDINATE\_SYSTEM\_ROTATION\_DEFAULT=angle\_value**

Outputs **Rangle\_value** on the **G68** line, where **angle\_value** is a specific angle, +/-180°. This might be useful when using jig fixtures that hold the part at the same angle.

## DATA\_DIR

The **DATA\_DIR** variable tells the **QUIKCODE**® software where to store the **Compustep**® part programs. The value of this variable should be in the form of:

d:\directory\

where:

**d:** is a valid drive letter

**\directory\** is the directory on the drive to store files in. Note that the directory must have a trailing backslash ("").

An example of a **DATA\_DIR** parameter is:

DATA\_DIR=C:\QC4\PARTS\

When the **QUIKCODE**® software is installed from the installation floppy disk, the **DATA\_DIR** is set to:

d:\install\_dir\PARTS\

where:

**d:\install\_dir** is the directory **QUIKCODE**® is installed in.

## DEFAULT\_NEGATIVE\_Z\_DEPTH

Determines if **QUIKCODE**® defaults to negative values for the depth in a drill or tap cycle.

**DEFAULT\_NEGATIVE\_Z\_DEPTH=0** **QUIKCODE**® will default to positive values for the depth in drill & tap cycles.

**DEFAULT\_NEGATIVE\_Z\_DEPTH=-1** **QUIKCODE**® will default to negative values for the depth in drill & tap cycles.

## DXF\_DIR

The **DXF\_DIR** variable tells **QUIKCODE**® where the CAD DXF files are stored. When the user selects DXF files in **QUIKCODE**®, all files with the .DXF extension in the **DXF\_DIR** directory are listed.

The **DXF\_DIR** variable should be in the form of:

d:\directory\

where:

**d:** a valid drive letter

**\directory\** is the directory on the drive that DXF files are stored in. Note that the directory must have a trailing backslash ("").

An example of a **DXF\_DIR** parameter is:

DXF\_DIR=C:\QC4\DXF\

When the **QUIKCODE**® software is installed from the installation floppy disk, the **DXF\_DIR** is set to:

d:\install\_dir\DXF\

where:

**d:\install\_dir** is the directory **QUIKCODE®** is installed in.

It may be preferable to change this variable to point to the CAD software's standard DXF directory. For instance, your CAD software may store DXF files in the directory **C:\CAD\DXF**. To change **QUIKCODE®** to use this DXF directory, change the **DXF\_DIR** variable to:

```
DXF_DIR=C:\CAD\DXF\
```

A drawback to this method is that the **QUIKCODE®** user will see all DXF files created by the CAD program, even those that are not destined to be converted into part programs. It may be safer or simpler to keep the DXF directory as it is installed and only write DXF files to that directory when they are to be converted to part programs.

## **FANCY\_SOUND**

The **FANCY\_SOUND** variable tells **QUIKCODE®** what type of sound to use when alerting the user. If **FANCY\_SOUND** is set to **Y**, **QUIKCODE®** will use more interesting sounds. If set to **N**, the standard console beep will be used. Set this variable to either:

```
FANCY_SOUND=Y
```

or

```
FANCY_SOUND=N
```

If **FANCY\_SOUND** is not set, or if it set to a value other than the ones shown, the standard console beep will be used.

## **FLOPPY\_COPY\_DRIVE**

The **FLOPPY\_COPY\_DRIVE** variable tells **QUIKCODE®** the drive letter to copy part programs to. This variable must be a single letter with no trailing colon (:) character. For the vast majority of installations, this variable will be set as:

```
FLOPPY_COPY_DRIVE=A
```

**QUIKCODE®** will use the **A:** floppy drive.

While this variable is intended to be a floppy drive, it is possible to set **FLOPPY\_COPY\_DRIVE** to any valid drive letter on the computer that **QUIKCODE®** is running on. If network drives or other media (removeable media such as ZIP drives, USB drives, memory cards, etc.) are mapped as drive letters on the computer, these can be specified. **QUIKCODE®** will write to the current working directory on the specified drive letter.

## **HOLE\_TYPE**

Sets the default entity type to search for in the CAD file (DXF or CDL). Valid entity types are **POINT** or **CIRCLE**. An example of the **HOLE\_TYPE** variable is:

```
HOLE_TYPE=CIRCLE
```

This variable should be set to the most common type of entity used in the CAD files.

## **MAX\_EXTENDED\_WORK\_OFFSET**

The **MAX\_EXTENDED\_WORK\_OFFSET** sets the largest extended offset value accepted by the CNC. Extended work offsets must be enabled (see **OFFER\_EXTENDED\_WORK\_OFFSETS** on page 54).

```
MAX_EXTENDED_WORK_OFFSETS=48
```

**QUIKCODE®** will accept extended offsets in the range 1 - 48.

## MAX\_RPM

The **MAX\_RPM** variable tells **QUIKCODE**® what the maximum spindle RPM is allowed on the machine. If a value larger than **MAX\_RPM** is entered when prompted for an RPM, **QUIKCODE**® will ignore the value and ask again. The format for setting **MAX\_RPM** is:

**MAX\_RPM=####**

where: **####** is the maximum RPM to allow.

For example:

**MAX\_RPM=3000**

sets the maximum RPM to 3000.

If not set, or if set to a nonsense values (negative values, for instance), the maximum RPM will be set to 9999 RPM.

## MAX\_TOOL\_TABLE\_NUM

The **MAX\_TOOL\_TABLE\_NUM** variable tells **QUIKCODE**® how large the tool length offset table on the CNC is. If a value larger than **MAX\_TOOL\_TABLE\_NUM** is entered when prompted for tool length offset, **QUIKCODE**® will ignore the value and ask again. The format for setting **MAX\_TOOL\_TABLE\_NUM** is:

**MAX\_TOOL\_TABLE\_NUM=###**

where: **###** is the maximum tool length offset table number.

For example:

**MAX\_TOOL\_TABLE\_NUM=99**

sets the maximum tool table offset to 99.

If not set, or if set to a nonsense values (negative values, for instance), the maximum tool table value will be set to 999.

Attempting to use a tool table value larger than is available on the CNC will cause an alarm and abort the part program.

## MIN\_RPM

The **MIN\_RPM** variable tells **QUIKCODE**® what the minimum spindle RPM is allowed on the machine. If a value smaller than **MIN\_RPM** is entered when prompted for an RPM, **QUIKCODE**® will ignore the value and ask again. The format for setting **MIN\_RPM** is:

**MIN\_RPM=####**

where: **####** is the minimum RPM to allow.

For example:

**MIN\_RPM=20**

Sets the minimum RPM to 20.

If not set, or if set to a nonsense values (negative values, for instance), the minimum RPM will be set to 1 RPM.

## OFFER\_FLOPPY\_COPY

The **OFFER\_FLOPPY\_COPY** variable tells **QUIKCODE**® to ask the user if the part program should be copied to a floppy disk drive after it is written to the normal output directory. Valid values for **OFFER\_FLOPPY\_COPY** are:

**OFFER\_FLOPPY\_COPY=0**      **QUIKCODE**® will not ask to copy the part program to floppy disk.

**OFFER\_FLOPPY\_COPY=-1**      **QUIKCODE**® will ask to copy the part program to floppy disk.

See the **FLOPPY\_COPY\_DRIVE** variable for how to set the drive letter for copying.

## OFFER\_M41

The **OFFER\_M41** variable tells **QUIKCODE**® to ask the user if layered drill cycle should be included with drill cycles. Since layered drill cycles are a purchased option on the CNC, customers who don't have this option on their CNC don't need to enter these codes. Valid settings for **OFFER\_M41** are:

**OFFER\_M41=0**      **QUIKCODE**® will not ask to include layered drill cycle codes in the part program.

**OFFER\_M41=-1**      **QUIKCODE**® will ask to include layered drill cycle codes in the part program.

Detailed instructions on how to enter layered drill cycle codes are described on page 39.

## OFFER\_EXTENDED\_WORK\_OFFSETS

The **OFFER\_EXTENDED\_WORK\_OFFSETS** variable tells **QUIKCODE**® to offer Fanuc-style extended offsets in the format G54.1 P###, where ### is the extended offset number.

**OFFER\_EXTENDED\_WORK\_OFFSETS=0**      **QUIKCODE**® will not offer extended offsets.

**OFFER\_EXTENDED\_WORK\_OFFSETS=-1**      **QUIKCODE**® will offer extended offsets.

## PROG\_DIR

The **PROG\_DIR** variable tells **QUIKCODE**® where it is installed on the hard drive. It must be in the form:

d:\directory\

This variable is automatically set when the **QUIKCODE**® software is installed. An example of the **PROG\_DIR** variable is (assuming **QUIKCODE**® was installed in the C:\QC4 directory):

PROG\_DIR=C:\QC4\

Note the trailing backslash (“\”) character.

## QM\_DATA\_DIR

The **QM\_DATA\_DIR** variable tells **QUIKCODE**® where to store part programs created for a **QUICKMILL/PC** or **QUICKDRILL/PC**. This directory should not be the same as other formats.

The value of the **QM\_DATA\_DIR** variable should be in the form of:

d:\directory\

where:

**d:** is a valid drive letter

**\directory\** is the directory on the drive to store files in. Note that the directory must have a trailing backslash ("").

An example of a **QM\_DATA\_DIR** parameter is:

```
QM_DATA_DIR=C:\QC4\QUIKMILL\
```

When the **QUIKCODE**® software is installed from the installation floppy disk, the **QM\_DATA\_DIR** is set to:

```
d:\install_dir\QUIKMILL\
```

where:

**d:\install\_dir** is the directory **QUIKCODE**® is installed in.

## **REALLY\_SLOW\_TIME**

The **REALLY\_SLOW\_TIME** variable tells **QUIKCODE**® how long to delay the plotting of holes when the **Left** and **Right s** keys are held down. The value is in seconds. The default is 0.5 seconds (1/2 a second).

## **RETURN\_TO\_R\_IN\_DRILL\_CYCLES**

The **RETURN\_TO\_R\_IN\_DRILL\_CYCLES** variable tells **QUIKCODE**® whether the tool should be returned to the **R** value or the initial level on retracting from a hole in a drill cycle.

**RETURN\_TO\_R\_IN\_DRILL\_CYCLES=0** the tool will be returned to the initial level when retracting from a hole. The initial level is the absolute coordinate of the tool axis (generally the Z axis) at the instant the canned drill cycle is called. The part program will contain a **G98 - Initial Level Return** command.

**RETURN\_TO\_R\_IN\_DRILL\_CYCLES=-1** the tool will be returned to the **R** value when retracting from a hole. The **R** value is specified on the canned drill cycle line. The part program will contain a **G99 - Return to R Value** command.

## **RIGID\_TAP\_SUPPORTED**

The **RIGID\_TAP\_SUPPORTED** tells **QUIKCODE**® whether rigid tapping is supported on the machine. If not supported, rigid tapping cycles are not offered by **QUIKCODE**®.

Valid settings for **RIGID\_TAP\_SUPPORTED** are:

**RIGID\_TAP\_SUPPORTED=0** **QUIKCODE**® will not offer rigid tap cycles.

**RIGID\_TAP\_SUPPORTED=-1** **QUIKCODE**® will offer rigid tap cycles.

## **SLOW\_TIME**

The **SLOW\_TIME** variable tells **QUIKCODE**® how long to delay the plotting of holes when the **Left s** key is held down. The value is in seconds. The default is 0.1 seconds (1/10th of a second).

## TEMP\_DIR

The **TEMP\_DIR** variable tells **QUIKCODE**® where to store its temporary files. These files are created **QUIKCODE**® operation. This directory must exist for **QUIKCODE**® to store various temporary files. This directory is created when **QUIKCODE**® is installed.

The value of the **TEMP\_DIR** variable should be in the form of:

d:\directory\

where:

**d**: a valid drive letter

**\directory\** is the directory on the drive to store files in. Note that the directory must have a trailing backslash ("").

An example of a **TEMP\_DIR** parameter is:

**TEMP\_DIR=C:\QC3\TEMP\**

When the **QUIKCODE**® software is installed from the installation floppy disk, the **TEMP\_DIR** is set to:

**d:\install\_dir\TEMP\**

where:

**d:\install\_dir** is the directory **QUIKCODE**® is installed in. There should be no reason to change this variable.

## UNITS

Sets the default units of measurement. Valid values are **INCH** and **METRIC**. This variable should be set to the most common units used in part programs. An example of the **UNITS** variable:

**UNITS=INCH**

## USE\_CYCLE\_PRELOAD

The **USE\_CYCLE\_PRELOAD** determines if **QUIKCODE**® will use an **L0** (for Fanuc 15i, Quickdrill/PC, or Quickmill/PC CNCs) or **K0** (for Intimidator/21i and B-Series machines) word on drill cycle lines to “pre-load” the drill cycle value but NOT drill a hole at the current location. The resulting part program will include a **GOTO** line immediately after the drill cycle line; the **GOTO** will direct the CNC to the first hole in the pattern.

**USE\_CYCLE\_PRELOAD=0**

The drill cycle pre-load method will not be used; the drill cycle line will not include an **L0** or **K0** word on the drill cycle line. Therefore, the CNC will execute a drill cycle at the current position when the drill cycle is called. It is imperative that the tool be placed in the correct location before the canned cycle line, as in the following example:

|                     |                                     |
|---------------------|-------------------------------------|
| N160 G55            | (SELECT WORK OFFSET)                |
| N170 G0 X+1.0 Y+2.0 | (RAPID TO FIRST HOLE)               |
| N180 G43 H2 G0 Z2.0 | (SELECT TOOL LENGTH, APPROACH PART) |
| N190 M3 S1000       | (SPINDLE ON)                        |
| N200 M11            | (FLOOD COOLANT ON)                  |
| N210 G99            | (RETURN TO R IN DRILL CYCLES)       |

```

N220 G81 Z-2.5 R0.5      (DRILL CYCLE)
N230 X+3.0 Y+2.0        (#2)
... (etc...)

```

Note that the tool is positioned above the first hole in line **N170**, since the CNC will perform a drill cycle at the current position when it is called in line **N220**.

#### USE\_CYCLE\_PRELOAD=-1

The drill cycle pre-load method will be used; the drill cycle line will include an **L0** or **K0** word on the drill cycle line. The CNC will not execute a drill cycle at the current position when the drill cycle is called; it will wait until after the next X and/or Y motion is complete. **QUIKCODE®** will insert a **GOTO** line immediately after the drill cycle line, as the following example shows:

```

N160 G55                  (SELECT WORK OFFSET)
N170 G0 X0.0 Y0.0        (RAPID TO PART REFERENCE)
N180 G43 H2 G0 Z2.0      (SELECT TOOL LENGTH, APPROACH PART)
N190 M3 S1000            (SPINDLE ON)
N200 M11                 (FLOOD COOLANT ON)
N210 G99                 (RETURN TO R IN DRILL CYCLES)
N220 G81 Z-2.5 R0.5 K0  (DRILL CYCLE)
N221 GOTO230            (JUMP TO FIRST HOLE)
N230 X+1.0 Y+2.0        (#1)
N240 X+3.0 Y+2.0        (#2)
... (etc...)

```

Note the **K0** word on line **N220**; this tells the CNC to setup the various arguments for the drill cycle (**R** word, **Z** word, as applicable for the specific drill cycle) but the CNC will not perform the drill cycle yet.

The **GOTO230** word in line **N221** causes the CNC to skip to line **N230**, which has the coordinates of the first hole; only after arriving at this position will the drill cycle be performed.

While it might seem pointless to have a **GOTO** command that simply skips to the next line, this method makes it trivial to restart the drilling operation at any hole by simply changing the **GOTO** line.

#### Undocumented Variables

There may be undocumented variables set in the configuration file. These are for special cases and can be ignored.



# CHAPTER 6

## INCLUDE FILES

---

### Purpose of Include Files

**QUIKCODE®** uses *include files* to allow the part programmer to modify the output part programs. Depending on the options on the CNC, shop standards, and personal preferences, the operator or part programmer may want to change the part program.

For instance, on machines without an Automatic Tool Changer (ATC), the operator may prefer the X & Y axes to always be placed in a convenient location for manual tool changes. By changing the **TC.INC** file (“tool change include file”) to have a move to **X=5, Y=5**, all part programs will have this line before a tool change.

### Changing Include Files

The include files are essential to the operation of **QUIKCODE®**. Extreme care should be taken when changing these files. The **QUIKCODE®** include files are unformatted ASCII text files and can be edited with an ASCII text file editor such as DOS **EDIT**, **EDLIN**, **Qedit**, etc. Do not use a word processor program to change the include files. Word processors add formatting codes to the file that **QUIKCODE®** will not be able to understand. This will cause unpredictable results from **QUIKCODE®**.

Approved editors include:

- DOS 5.x and higher: **EDIT**
- DOS 4.x and lower: **EDLIN**

Other editors may work. Please contact **QUICKMILL Inc.** to determine if your editor will work.

The **QUIKCODE®** include files are stored in the same directory as the **QUIKCODE.EXE** file. There are three include files, depending on the type of CNC:

#### **QUICKMILL/PC or QUICKDRILL/PC CNC**

- **PRE\_PROG.INC**
- **TC.INC**
- **END\_PROG.INC**

#### **QUICKMILL/15i CNC**

- **15I\_PRE.INC**
- **15I\_TC.INC**
- **15I\_END.INC**

#### **Intimidator/21i CNC**

- **21I\_PRE.INC**
- **21I\_TC.INC**
- **21I\_END.INC**

## QUICKMILL/QUICKDRILL Controller

### PRE\_PROG.INC

The **PRE\_PROG.INC** include file is added to the part program immediately following the code that sets the current units (**G20** or **G21**). **PRE\_PROG.INC** contains lines of code that should be included at the start of every part program.

A typical example of a **PRE\_PROG.INC**:

```
1:  G80                      (CANCEL CANNED CYCLES)
2:  G28 X0 Y0 Z0 H0         (CANCEL ALL OFFSETS)
3:  T000                    (SEND 0 TO ATC)
```

(The lines numbers (1:, 2:, etc.) are for illustration only and are not part of the include file.)

### TC.INC

The **TC.INC** include file is added to the part program before the first tool is used and whenever a tool is changed thereafter. A typical **TC.INC** file is:

```
1:  G80                      (CANCEL CANNED CYCLE)
2:  M05 M09                  (SPINDLE OFF, COOLANT OFF)
3:  G28 X0 Y0 Z0 H0         (CANCEL ALL OFFSETS)
4:  G00 Z0                   (RAISE Z TO CLEAR)
5:  G00 X5 Y5                (MOVE XY TO T/C POSITION)
6:  T### H###                (USE TOOL, LENGTH OFFSET)
7:  M06                      (TOOL CHANGE)
```

(The lines numbers (1:, 2:, etc.) are for illustration only and are not part of the include file.)

### END\_PROG.INC

The **END\_PROG.INC** include file is added to the end of the part program. A typical **END\_PROG.INC** might be:

```
1:  G80                      (CANCEL CANNED CYCLES)
2:  M05 M09                  (SPINDLE OFF, COOLANT OFF)
3:  G28 X0 Y0 Z0             (CANCEL ALL OFFSETS)
4:  T000 H000                (CANCEL TOOL OFFSETS)
5:  M06
6:  ?M02                     (END OF PROGRAM)
7:  ?M30                     (END OF PROGRAM, REWIND)
8:  %
```

## QUICKMILL/15i

### 15I\_PRE.INC

The **15I\_PRE.INC** include file is added to the part program immediately following the code that sets the current units (**G20** or **G21**). **15I\_PRE.INC** contains lines of code that should be included at the start of every part program. A typical example of a **15I\_PRE.INC**:

```
1:  G80 G17 G80 G43 H0 (CLEAR EVERYTHING)
```

(The lines numbers (1:, 2:, etc.) are for illustration only and are not part of the include file.)

### 15I\_TC.INC

The **15I\_TC.INC** include file is added to the part program before the first tool is used and whenever a tool is changed thereafter. A typical **15I\_TC.INC** file is:

```
1:  G80                      (CANCEL CANNED CYCLE)
2:  M05                      (SPINDLE OFF)
3:  M09                      (COOLANT OFF)
4:  G53 H0 G0 Z0            (RAPID Z TO 0)
5:  M06 T###                (CHANGE TOOL)
6:  G43 H###                (USE TOOL, LENGTH OFFSET)
```

(The lines numbers (1:, 2:, etc.) are for illustration only and are not part of the include file.)

### 15I\_END.INC

The **15I\_END.INC** include file is added to the end of the part program. A typical **15I\_END.INC** might be:

```
1:  G80                      (CANCEL CANNED CYCLES)
2:  M05                      (SPINDLE OFF)
3:  M09                      (COOLANT OFF)
4:  ?M02                    (END OF PROGRAM)
5:  ?M30                    (END OF PROGRAM AND REWIND)
```

(The lines numbers (1:, 2:, etc.) are for illustration only and are not part of the include file.)

## Intimidator/21i

### 21I\_PRE.INC

The **21I\_PRE.INC** include file is added to the part program immediately following the code that sets the current units (**G20** or **G21**). **21I\_PRE.INC** contains lines of code that should be included at the start of every part program.

A typical example of a **21I\_PRE.INC**:

```
1:  G80 G17 G43 G49 G69 H0 (CLEAR EVERYTHING)
```

(The lines numbers (1:, 2:, etc.) are for illustration only and are not part of the include file.)

## 21I\_TC.INC

The **21I\_TC.INC** include file is added to the part program before the first tool is used and whenever a tool is changed thereafter.

A typical **21I\_TC.INC** file is:

```
1:  G80                      (CANCEL CANNED CYCLE)
2:  M05                      (SPINDLE OFF)
3:  M09                      (ALL COOLANT OFF)
4:  G53 H0 G0 Z0.           (RAPID Z TO 0)
5:  M06 T###                (CHANGE TOOL)
6:  G54 P###                (SELECT WORK OFFSET)
7:  ?G68 X#.# Y#.# R#.#    (COORDINATE SYSTEM ROTATION)
8:  G00 X#.# Y#.#          (RAPID TO START)
9:  G43 H### G00 Z#.#      (ADD TOOL LENGTH, APPROACH PART)
```

(The lines numbers (1:, 2:, etc.) are for illustration only and are not part of the include file.)

## 21I\_END.INC

The **21I\_END.INC** include file is added to the end of the part program.

A typical **21I\_END.INC** might be:

```
1:  G80                      (CANCEL CANNED CYCLES)
2:  M05                      (SPINDLE OFF)
3:  M09                      (COOLANT OFF)
4:  ?M02                    (END OF PROGRAM)
5:  ?M30                    (END OF PROGRAM AND REWIND)
```

(The lines numbers (1:, 2:, etc.) are for illustration only and are not part of the include file.)

## B-Series

### B18\_PRE.INC

The **B18\_PRE.INC** include file is added to the part program immediately following the code that sets the current units (**G20** or **G21**). **B18\_PRE.INC** contains lines of code that should be included at the start of every part program.

A typical example of a **B18\_PRE.INC**:

```
1:  G80 G17 G43 G49 G69 H0  (CLEAR EVERYTHING)
```

(The lines numbers (1:, 2:, etc.) are for illustration only and are not part of the include file.)

### B18\_TC.INC

The **B18\_TC.INC** include file is added to the part program before the first tool is used and whenever a tool is changed thereafter.

A typical **B18\_TC.INC** file is:

```
1:  G80                      (CANCEL CANNED CYCLE)
2:  T###                    (SELECT NEXT TOOL)
```

```

3:    M05                      (SPINDLE OFF)
4:    M09                      (ALL COOLANT OFF)
5:    G53 H0 G0 Z0.           (RAPID Z TO 0)
6:    M06                      (CHANGE TOOL)
7:    G54 P###                (SELECT WORK OFFSET)
8:    G68 X#.# Y#.# R#.#      (COORDINATE SYSTEM ROTATION)
9:    G00 X#.# Y#.#           (RAPID TO START)
10:   G43 H### G00 Z#.#       (ADD TOOL LENGTH, APPROACH PART)

```

(The lines numbers (1:, 2:, etc.) are for illustration only and are not part of the include file.)

## **B18\_END.INC**

The **B18\_END.INC** include file is added to the end of the part program.

A typical **B18\_END.INC** might be:

```

1:    G80                      (CANCEL CANNED CYCLES)
2:    M05                      (SPINDLE OFF)
3:    M09                      (COOLANT OFF)
4:    ?M30                     (END OF PROGRAM AND REWIND)

```

(The lines numbers (1:, 2:, etc.) are for illustration only and are not part of the include file.)

## **Variable Coordinates (X#.#, Y#.#, Z#.#)**

A variable coordinate can be specified in and include file, in the format:

**X#.#** - an X axis variable coordinate

**Y#.#** - a Y axis variable coordinate

**Z#.#** - a Z axis variable coordinate

When **QUIKCODE®** encounters a variable coordinate, it will ask the user for the coordinate value with a prompt such as:

**Enter the X value for this line: \_\_**

Enter the required coordinate value. If nothing is entered, **QUIKCODE®** will leave the variable coordinate on the line, which will have to be corrected before attempting to run the part program. This may be useful in situations where the machine operator is required to set certain coordinates instead of the part programmer.

## **Optional Lines**

A line in any of the include files can be made optional by adding the question mark “?” character to the start of the line. In the **15I\_END.INC** file above, lines 4 and 5 are optional. When **QUIKCODE®** encounters a optional line, it will ask the user if the line should be included in the part program. In this example, the user can chose to have the part program end with an **M02** code to stop the program and exit, or an **M30** code, to stop the program and rewind to the beginning. If the programmer and/or operator decide that all part programs should end with an **M30** command, then the **15I\_END.INC** file can be edited to remove the optional **M02** line and “hard code” the **M30** command. Such a file might look like:

|    |      |                             |
|----|------|-----------------------------|
| 1: | G80  | (CANCEL CANNED CYCLES)      |
| 2: | M05  | (SPINDLE OFF)               |
| 3: | M09  | (COOLANT OFF)               |
| 4: | ?M02 | (END OF PROGRAM)            |
| 5: | ?M30 | (END OF PROGRAM AND REWIND) |

